Numerical Predictions of a Model Scale Propeller in Uniform and Oblique Flow

Mitja MORGUT [a,1], Dragica JOŠT [b] Aljaž ŠKERLAVAJ [b] Enrico NOBILE [a]
Giorgio CONTENTO [a] Riccardo PIGAZZINI [a] Thomas PUZZER [a] Simone MARTINI [a]

[a] *Università degli Studi di Trieste, via A. Valerio 10, 34127 Trieste, Italy*
[b] *Kolektor-Turboinštitut, Rovšnikova 7, 1210 Ljubljana, Slovenia*

Abstract. The numerical predictions of a model scale propeller working in uniform and oblique flow are presented. Both non-cavitating and cavitating flow conditions are numerically investigated using homogeneous (mixture) model. Two previously calibrated mass transfer models are alternatively used to model the mass transfer rate due to cavitation. The turbulence effect is modelled using the Reynolds Averaged Navier Stokes (RANS) approach. The simulations are performed using an open source solver. The numerical results are compared with the available experimental data. For a quantitative comparison the propeller thrust is considered, while for a qualitative comparison, snapshots of cavitation patterns are shown. From the overall results it seems that, with the current simulation approach it is possible to predict with a reasonable accuracy the propeller performances. Nevertheless, for detailed reproduction of complex cavitation phenomena, such as bubble cavitation for instance, a more sophisticated modelling approach is probably required.

Keywords. Propeller, cavitation, uniform inflow, oblique flow, URANS

1. Introduction

Over the last decades due to the steady improvement of Computational Fluid Dynamics (CFD) technologies as well as computer performances, numerical simulations have become a valuable and reliable tool for design purposes, allowing, in general, the more expensive and time consuming experimental tests to be performed only at the final stages of the project.

In the specific case of marine propellers, CFD analysis can be effectively used to predict overall machine performances as well as to investigate the effects of particular flow phenomena such as cavitation.

Here, the numerical predictions of a model scale propeller working in uniform and oblique flow are presented.

The current results were obtained using the homogeneous mixture model along with two different cavitation (mass transfer) models. More precisely, the models originally proposed by Kunz et al. [1] and Zwart et al. [2], and modified according to [3,4] were used. The turbulence effect was modelled using the standard RANS approach in combination

---

with the Shear Stress Transport (SST) turbulence model [5].

All the simulations were carried out using the *interPhaseChangeDyMFoam* solver available in the OpenFOAM-4.1 [6]

The numerical results were compared with the experimental data. For both uniform and oblique flow conditions a quantitative comparison was performed considering the propeller thrust. For an additional qualitative comparison, useful to justify the predicted thrust values, sketches/snapshots of cavitation patterns were used.

The numerical results were in line with the experimental data. The effect of cavitation on the delivered thrust was well reproduced, even though some discrepancies were observed in predicted cavitation patterns. In particular, with the current modelling approach, the bubble cavitation present on the propeller blades in the case of oblique flow conditions was not properly captured. Nevertheless, it is worth pointing out that the differences between the simulations performed using alternatively the two different cavitation models were minimal.

Below, the test case is described and the mathematical model is briefly presented followed by the numerical strategy used to perform the simulations. The results obtained for the uniform and oblique flow conditions are discussed. Finally, some concluding remarks are given.

2. Test Case

In this study the PPTC (Potsdam Propeller Test Case) model propeller was used as a reference test case. The propeller in question is a five-bladed, controllable pitch propeller with a diameter D_p=0.250 m. For this propeller uniform and oblique flow conditions were numerically investigated during the *Workshop on Cavitation and Propeller Performance* in 2011 [7] and 2015 [8], respectively. A significant amount of experimental data, useful to validate CFD predictions, is currently available at [9]. Here, a limited number of operating regimes is discussed.

3. Mathematical Model

Cavitating flows can be modelled using several methods. A review of different methods is for instance provided by [10]. The homogeneous transport equation used in this study was already successfully employed in our former studies [11,12]. It is based on the continuity and momentum equations for the liquid-vapour mixture and an additional transport equation for liquid volume fraction. In the continuity and volume fraction equations appropriate source terms are present in order to control the mass transfer rate between the two phases. These source terms can be modelled using different mass transfer models. In this study two previously calibrated models were used. In particular the models originally proposed by Kunz et al. and Zwart et al., where the empirical coefficients were modified according to [3], were used. At this stage, it is worth clarifying that in the case of RANS simulations additional transport equations equation have to be solved according to the selected turbulence model. In our case the transport equations for the turbulent kinetic energy, k, and turbulent frequency, ω, implemented in the Shear stress Turbulence (SST)

model were solved. For further details regarding the homogeneous model as well as its implementation in OpenFOAM we refer to [13] for convenience.

4. Numerical setup

The numerical simulations were performed following the experimental setup. In current simulations the computational domains, roughly sketched in Fig. 1, were used for the cases of the propeller in uniform and oblique flow, respectively. In the case of the oblique flow, the propeller according to experimental setup, had an incidence inclination of 12 degrees towards the inflow direction.
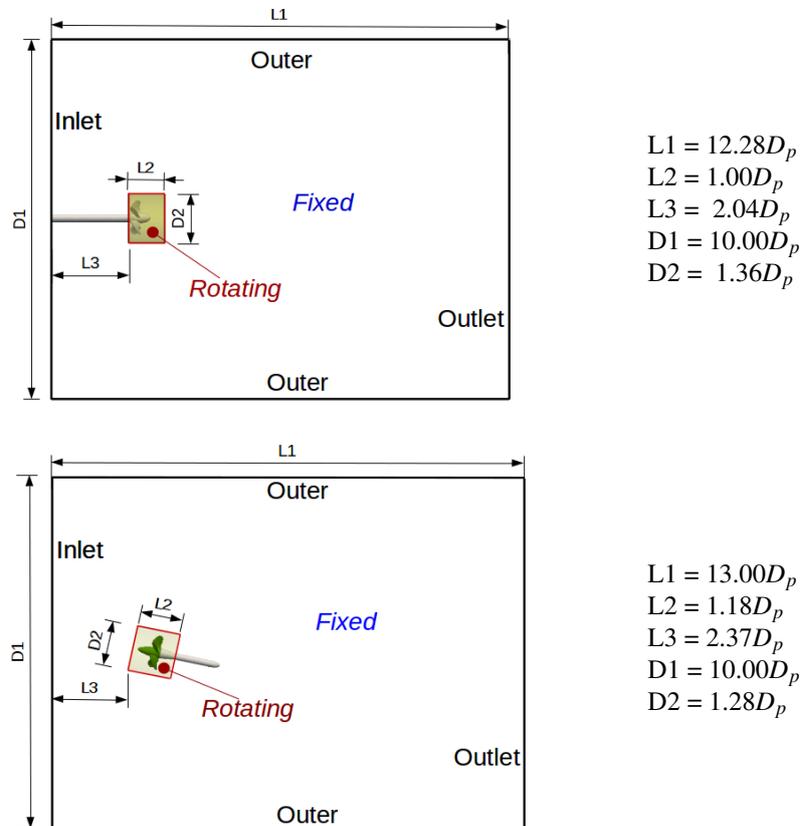
Figure 1. Sketches of the computational domains used to numerically investigate the PPTC propeller working in uniform (top), and inclined flow (bottom). Both *Rotating* and *Fixed* regions were cylinders.

The propeller rotation was simulated using the *dynamic mesh motion* capabilities of the *interPhaseChangeDyMFoam* solver. Thus, the computational domains were subdivided into *Rotating* and *Fixed* mesh/domain regions.

For time discretization, a first order implicit time scheme was used, while for the discretization of the advective terms a second order *linearUpwind* scheme was employed.

For turbulence closure the two-equation SST turbulence model was used in combination with the automatic wall treatment available in OpenFOAM [6].

For all the different cavitating flow regimes the two different calibrated mass transfer models were alternatively used.

The boundary conditions, summarized in Table 1, were imposed.

Table 1. Boundary conditions common for both uniform and oblique flow conditions

Boundary	V (m/s)	*Pressure* (Pa)	$k \; (m^2/s^2)$	ω (1/s)
Inlet	fixedValue	zeroGradient	fixedValue = 0.06	fixedValue = 2000
Outlet	zeroGradient	fixedValue = 101325	zeroGradient	zeroGradient
Outer	slip	zeroGradient	zeroGradient	zeroGradient
Propeller	movingWallVelocity	zeroGradient	kqRWallFunction	omegaWallFunction

It is worth clarifying that the different cavitating flow regimes were identified by the advance coefficient, J, and cavitation number σ_n defined as follows:

$$J = \frac{V}{nD_p} \qquad \sigma_n = \frac{P_{Outlet} - Pv}{0.5\rho(nD_p)^2}$$

with n (rps) being propeller rotation and Pv (Pa) vapour pressure. Thus, the velocity imposed on Inlet boundary and the value of the vapour pressure were varied according to J and σ_n.

As regards the computational meshes, in this study, the overall computational grids had approximately 3,500,000 cells for both cases. In Fig. 2 details of surface meshes are shown. The corresponding y^+ value was approximately equal to 25 for both meshes.
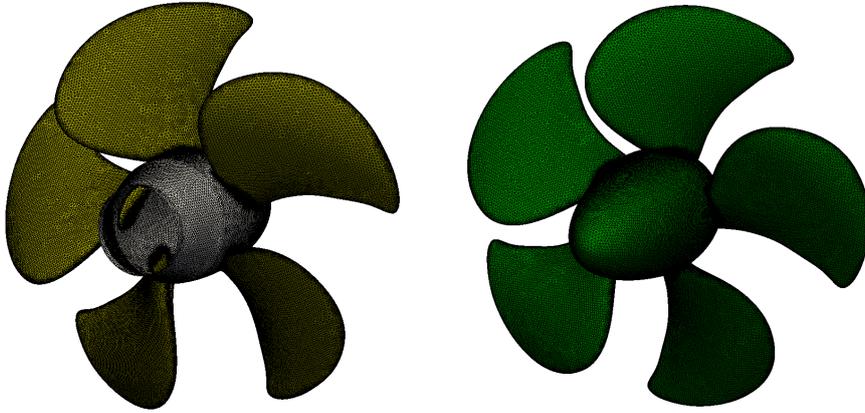
Figure 2. Surface meshes for uniform inflow (left) and oblique (right) flow.

5. Results

Here, the numerical results are compared with experimental data. The values of the thrust coefficient $K_T = Tx/(\rho n^2 D_p^4)$, defined considering the propeller thrust in direction of the rotational axis, T_x, is evaluated for a quantitative comparison. For further qualitative comparison, snapshots of cavitation patterns are depicted.

5.1. Uniform inflow

Four different operating conditions were numerically predicted. Both *fully wetted* and cavititating flow regimes were considered. The simulations were performed according to the experimental setup.

Table 2. Thrust values for the uniform inflow condition

J	σ_n	K_T (Exp.)	K_T (Kunz)	K_T (Zwart)
1.019	(non cavitating)	0.387	0.375	0.375
1.019	2.024	0.372	0.377	0.377
1.408	(non cavitating)	0.167	0.161	0.161
1.408	2.000	0.136	0.142	0.143

$$J = 1.019, \sigma_n = 2.024 \qquad J = 1.408, \sigma_n = 2.000$$
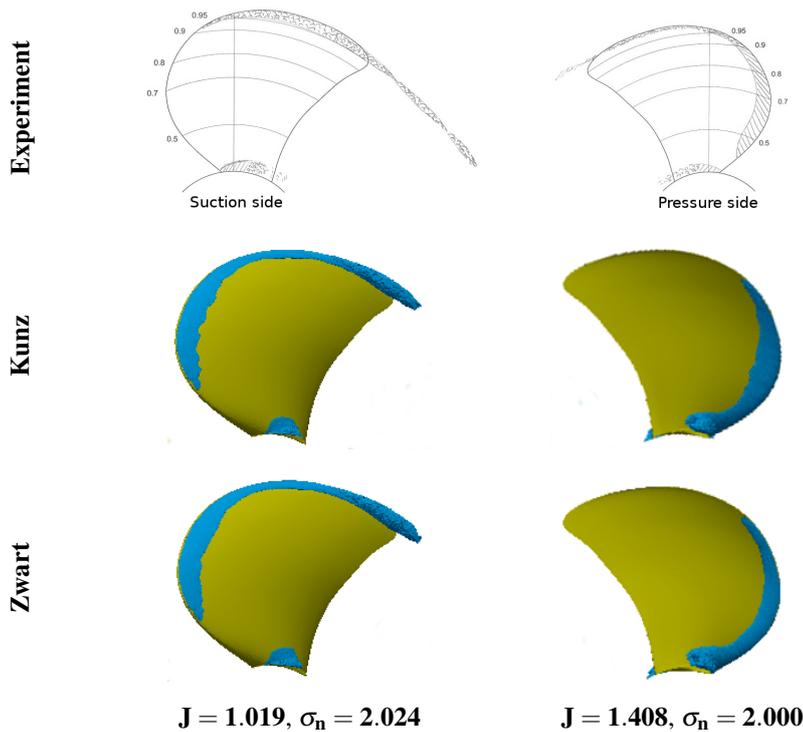
Figure 3. Comparison of the cavitation patterns for uniform inflow. The numerical cavitation patterns, obtained using the two different calibrated mass transfer models, are depicted as isosurfaces of vapour volume fraction $\alpha = 0.2$. Experimental sketches adapted from [7].

From Table 2 it is possible to note that the overall numerical results compared reasonably well with experimental data. The differences between numerical and experimental values were within the range of 5%. The effect of cavitation on propeller thrust was reproduced, however, at $J = 1.019$ and for fully wetted flow conditions, the thrust was un-
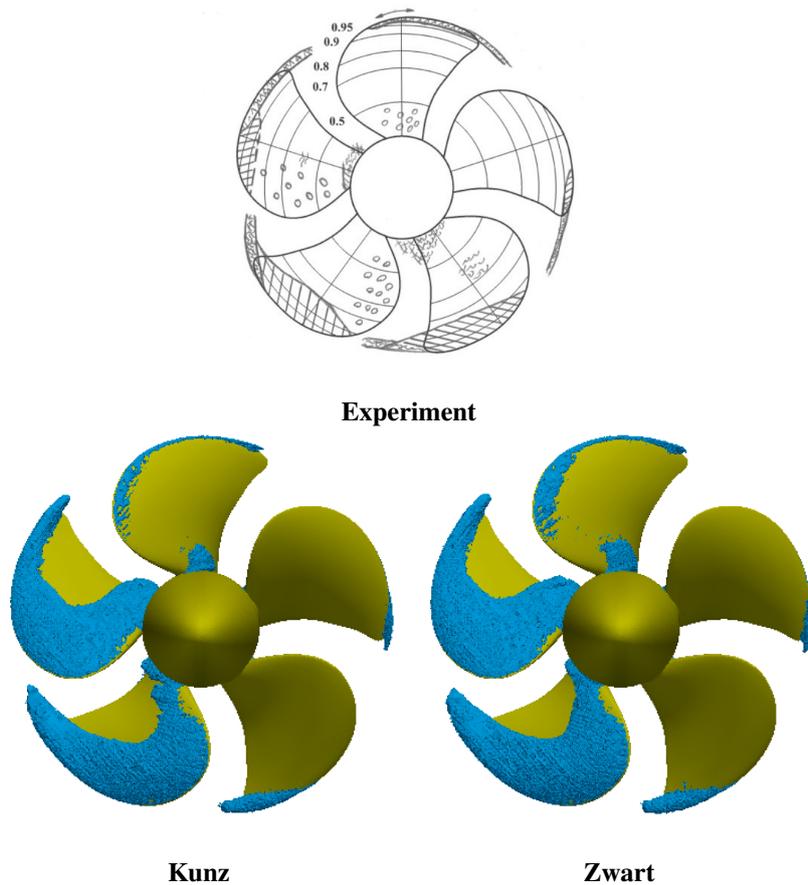
derestimated. The thrust values predicted using the two different calibrated mass transfer models were very close. As a matter of fact, the cavitation patterns obtained using the two different mass transfer models alternatively were very similar as qualitatively shown in Fig. 3

5.2. Oblique flow

In this case the simulations were carried out only for one value of the advance coefficient following the indications given in [9] .

Table 3. Thrust values for oblique flow conditions

J	σ_n	K_T (Exp.)	K_T (Kunz)	K_T (Zwart)
1.019	(non cavitating)	0.392	0.411	0.411
1.019	2.024	0.363	0.387	0.383

Figure 4. Comparison of the cavitation patterns at inclined flow for $J = 1.019$, $\sigma_n = 2.000$. The numerical cavitation patterns, obtained using the two different calibrated mass transfer models, are depicted as isosurfaces of vapour volume fraction $\alpha = 0.2$. Experimental data from [8].

Table 3 shows the thrust coefficients predicted for both non-cavitating and cavitating flow conditions. It is possible to note that also in this case the numerical results obtained using the two different mass transfer models alternatively were very similar and compared reasonably well with the experimental data. For both non cavitating and cavitating flow regimes the thrust predicted by the CFD simulations was overestimated.

In Fig. 4, for the sake of completeness, cavitation patterns obtained at a certain (istantaneous) propeller position are shown. We can see that in this preliminary study by employing a rather simple model the numerical simulations were not capable of properly reproducing the phenomenon of bubble cavitation experimentally observed close to the propeller leading edge. In the case of the numerical simulations this zone was covered by sheet cavitation.

6. Conclusions

In this study the numerical predictions of the PPTC model propeller working in uniform and oblique flow conditions were discussed. The cavitating flow was reproduced using the homogeneous model. Two different mass transfer models were employed in order to reproduce cavitation. The turbulence effect was taken into account using the RANS approach in combination with the SST turbulence model.

The numerical simulations were carried out using the *interPhaseChangeDyMFoam* solver present in OpenFOAM-4.1

The numerical results were compared with the available experimental data. From the overall results, it emerges that using the proposed numerical strategy the performance of a propeller in both uniform and oblique flow configurations can be reasonably predicted. The effect of cavitation on the delivered thrust can be reproduced well; however, a more advanced modelling approach should probably be used for an improved reproduction of the local structures of the cavitation phenomenon.

Acknowledgment

References

[1] Kunz, R. F., Boger, D. A., Stinebring, D. R., Chyczewski, T. S., Lindau, J. W., Gibeling, H. J., Venkateswaran, S., Govindan, T. R. A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction. *Computers and Fluids*, 29 (8), pp. 849-875, 2000. DOI:10.1016/S0045- 7930(99)00039-0.

[2] Zwart, P. J., Gerber, A. G., Belamri, T. A Two-Phase Model for Predicting Cavitation Dynamics, *ICMF 2004 International Conference on Multiphase Flow, Yokohama, Japan*, 2004

[3] Morgut, M., Jošt, D., Nobile, E., Škerlavaj, A. Numerical investigations of a cavitating propeller in non-uniform inflow. *In proceedings of: Fourth International Symposium on Marine Propulsors, smp'15*, Austin, Texas, USA, June 2015.

[4] Morgut, M., Nobile, E. Numerical Predictions of Cavitating Flow around Model Scale Propellers by CFD and Advanced Model Calibration. *International Journal of Rotating Machinery*, Volume 2012, Article ID 618180, 11 pages. DOI:10.1155/2012/618180.

[5] Menter, F. R. Two-equation eddy-viscosity turbulence model for engineering applications, *AIAA Journal*, 32(8), pp. 1598-1605, 1994. DOI:10.2514/3.12149.

[6] www.openfoam.org

[7] *Proceedings of the Workshop on Cavitation and Propeller Performance, The Second International Symposium on Marine Propulsors, smp'11*, 17-18 June, 2011, Hamburg, Germany. Proceedings available at: http://www.marinepropulsors.com/proceedings-2011.php.

[8] *Proceedings of the Second Workshop on Cavitation and Propeller Performance, The Fourth International Symposium on Marine Propulsors, smp'15*, June 4, 2015, Austin, Texas, USA. Proceeding available at: http://www.marinepropulsors.com/proceedings/2015/2nd-Workshop-smp15.pdf.

[9] https://www.sva-potsdam.de/en/potsdam-propeller-test-case-pptc/

[10] Koop, A. H. Numerical Simulation of Unsteady Tree- Dimensional Sheet Cavitation. *PhD thesis. University of Twente*, Twente, 2008.

[11] Morgut, M., Jošt, D., Nobile, E., Škerlavaj, A. Numerical predictions of the turbulent cavitating flow around a marine propeller and an axial turbine. *Journal of Physics: Conference Series*, 656, 2015. DOI:10.1088/1742-6596/656/1/012066.

[12] Morgut, M., Jošt D., Škerlavaj, A., Nobile, E., Contento, G. Numerical Predictions of Cavitating Flow Around a Marine Propeller and Kaplan Turbine Runner with Calibrated Cavitation Models. *Strojniški vestnik - Journal of Mechanical Engineering* 64 (2018), pp. 1-12. DOI:10.5545/sv-jme.2017.4647. (Article in Press).

[13] Gaggero, S., and Villa, D. Steady cavitating propeller performance by using OpenFOAM, StarCCM+ and a boundary element method. *Journal of Engineering for the Maritime Environment* Vol. 231(2) 41144, 2017. DOI:10.1177/1475090216644280.