Numerical predictions of the turbulent cavitating flow around a marine propeller and an axial turbine

Numerical predictions of the turbulent cavitating flow around a marine propeller and an axial turbine

M Morgut[1], D Jošt[2], E Nobile[3] and A Škerlavaj[2]

[1] Turboinštitut d.d., Ljubljana, Slovenia

[2] Turboinštitut d.d., Ljubljana, Slovenia, presently at University of Trieste, Department of Engineering and Architecture, Italy

[3] University of Trieste, Department of Engineering and Architecture, Italy

E-mail: mitja.morgut@kolektor.com

Abstract. The numerical predictions of cavitating flow around a marine propeller working in non-uniform inflow and an axial turbine are presented. The cavitating flow is modelled using the homogeneous (mixture) model. Time-dependent simulations are performed for the marine propeller case using OpenFOAM. Three calibrated mass transfer models are alternatively used to model the mass transfer rate due to cavitation and the two-equation SST (Shear Stress Transport) turbulence model is employed to close the system of the governing equations. The predictions of the cavitating flow in an axial turbine are carried out with ANSYS-CFX, where only the native mass transfer model with tuned parameters is used. Steady-state simulations are performed in combination with the SST turbulence model, while time-dependent results are obtained with the more advanced SAS (Scale Adaptive Simulation) SST model. The numerical results agree well with the available experimental measurements, and the simulations performed with the three different calibrated mass transfer models are close to each other for the propeller flow. Regarding the axial turbine the effect of the cavitation on the machine efficiency is well reproduced only by the time dependent simulations.

1. Introduction

Recently the University of Trieste (Italy) and Turboinštitut d.d. from Ljubljana (Slovenia) joined in the ACCUSIM-EU project that aims, primarily, to develop reliable, high fidelity methods for the accurate predictions, and optimization, of the performances of hydro-machinery and marine propellers.

In reference to this project, in this work selected results obtained from this successful collaboration are presented for a marine propeller and an axial turbine. In particular, the numerical investigations of the CNR-INSEAN E779A model propeller are presented, and the simulations performed considering a 6-blade Kaplan turbine are discussed.

In current predictions the turbulent cavitating flow was simulated using the so called homogeneous model. This model treats the working fluid as a homogeneous mixture of two fluids, i.e water and vapour, behaving as a single one, and the mass transfer rate due to cavitation is regulated by the mass transfer model. In the literature it is possible to find several mass transfer models. In this work the models originally proposed by Kunz et al. [1], Singhal et al. [2], Zwart et al. [3] and calibrated as described in [4] were considered.

In the case of the marine propeller working in non-homogeous (in wake) inflow conditions, the simulations were performed using OpenFOAM, an open source CFD (Computational Fluid Dynamics)

toolbox [5]. Time-dependent simulations were carried out using alternatively all the three different calibrated mass transfer models in combination with the SST turbulence model.

As far as the Kaplan case is concerned, the predictions were performed using ANSYS-CFX 15, a commercial CFD code. Steady-state and time-dependent predictions were carried out using the SST turbulence model and the SST-SAS method, respectively. In both cases the curvature correction and Kato-Launder limiter, for the production in the turbulent kinetic energy equation, were included following previous studies [6]. The mass transfer rate due to cavitation was evaluated using the native Zwart et al. mass transfer model with tuned empirical coefficients [3, 4].

2. Meshing

The meshes used in the current simulations were generated using ANSYS-ICEM CFD 15, a commercial meshing tool. In figure 1 two snapshots of the grids generated for the propeller and axial turbine are presented.
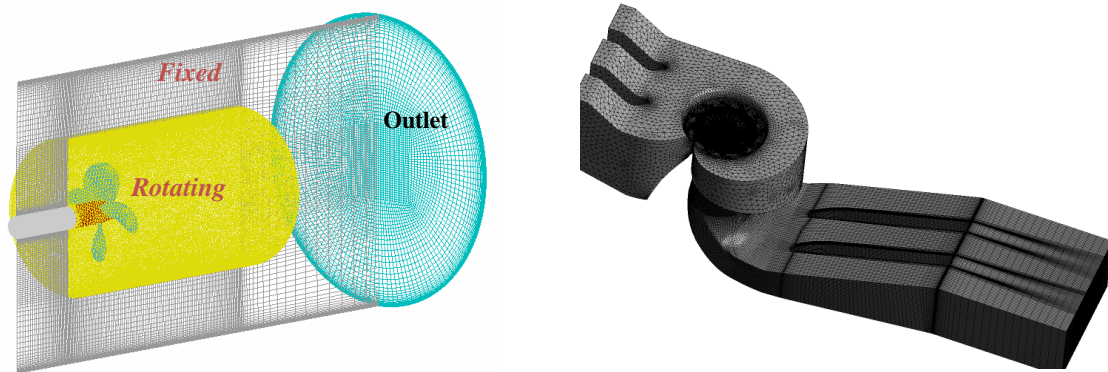

Figure 1. Computational grids for the marine propeller (left) and Kaplan turbine (right).

The mesh for the propeller simulations was generated using two different approaches. The rotating region (in yellow in figure 1) was created following the hybrid approach (tetrahedral + prisms layers at the walls) while the fixed region was discretized by the hexa-structured grid. The overall mesh had about 8.1 million cells. Regarding the Kaplan turbine a hybrid mesh was generated for the semi-spiral casing with stay vanes, while for the guide vane cascade, runner and elbow draft tube hexa-structured meshes were generated. The overall mesh had about 8.3 million nodes. All the grids were refined at the walls according to the recommended values of y^+.

3. Numerical simulations

3.1 E779A propeller

The numerical simulations were carried out following the experimental/numerical setup described in [7]. In particular, here we point out that on the domain inlet the non-homogeneous inflow (nominal wake), kindly provided by CNR-INSEAN (private communication) was set. On outlet boundary a fixed value of static pressure was imposed. On the solid surfaces the no-slip wall condition was enforced. A second order upwind scheme was adopted for the discretisation of the convective terms while a first order implicit scheme was used for the time discretization.

From figure 2 it is possible to note that a similar cavity evolution was predicted using alternatively the three different mass transfer models. As can be seen, the overall numerical predictions agree well with the available experimental data.
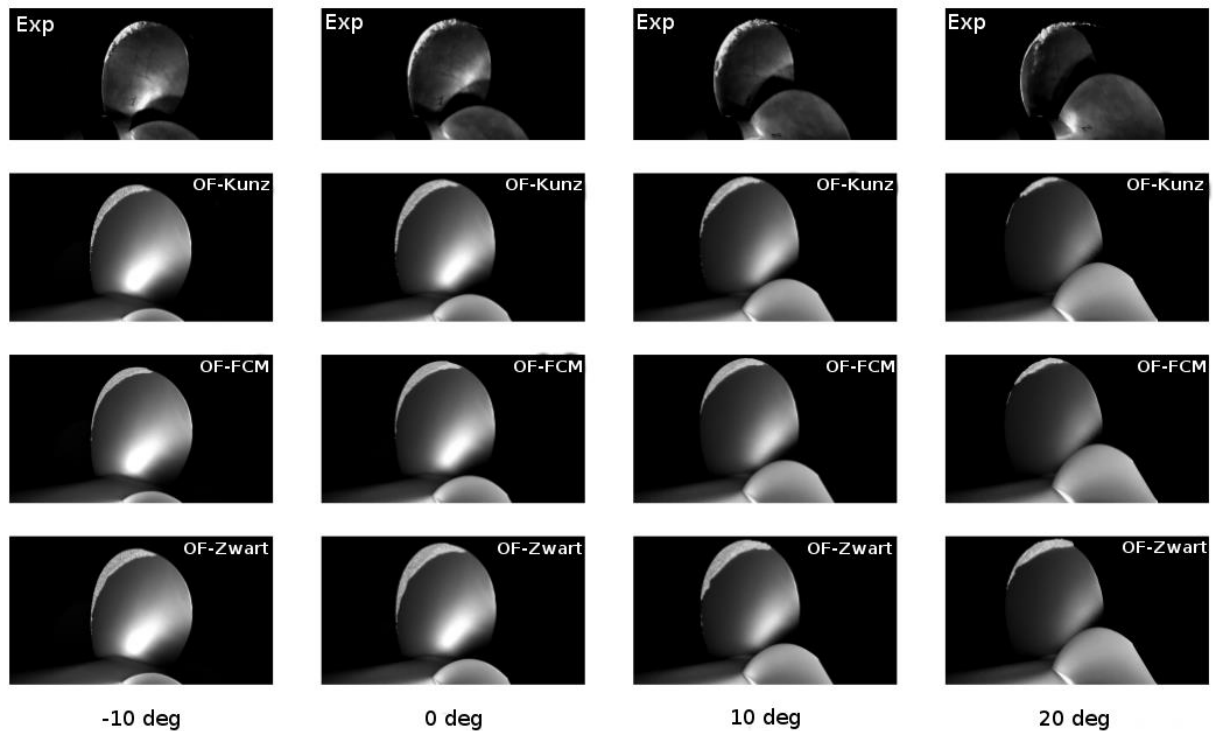
Figure 2. Cavity evolution during propeller rotation. Numerical cavitation patterns depicted using isosurfaces of vapour volume fraction equal to 0.1. In the above figures OF stays for OpenFOAM, while FCM stays for Full Cavitation Model [2].

3.2 Kaplan turbine

In order to determine the effect of cavitation on turbine efficiency (sigma-break curve) successive simulations were performed lowering the reference pressure. Steady state simulations were performed using the MRF (Multiple Reference Frame) approach in combination with the *frozen rotor* frame/change mixing model. Time-dependent simulations were performed using *sliding grids*. In both cases the robust high resolution scheme was used for the discretization of the convective terms.
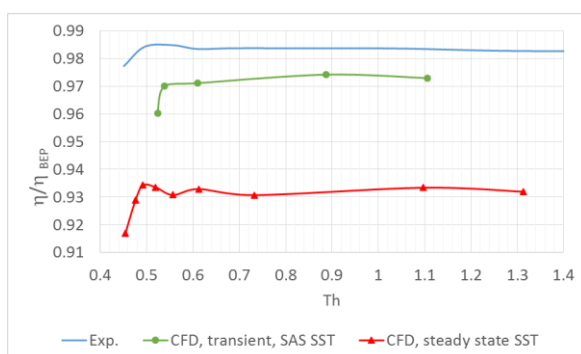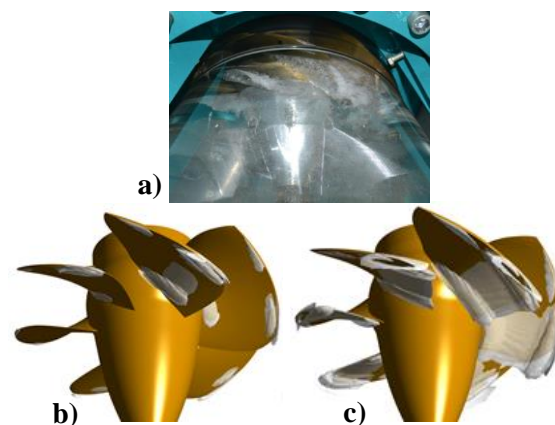

Figure 3. Sigma break curve

Figure 4. Cavity on runner blades: a) observation on the test rig, b) steady state simulation, c) time dependent simulation ; Th=0.52

Numerical results were compared with the observation of cavity size on the test rig and with the measured sigma-break curve. Steady-state simulations did not predict the same amount of cavitation on all blades due to frozen rotor assumption, which preserved differences in circumferential direction. Besides, the extent of cavitation was too small compared to the experimental one, as illustrated in figure 4.

With transient simulations the same amount of cavitation on all runner blades was obtained and the shape and extent of sheet cavitation agreed well with the cavitation observed on the test rig (figure 4). Steady-state simulations significantly underpredicted the efficiency, as illustrated in figure 3. Transient simulations predicted the efficiency more accurately, although the Thoma number, Th, where the efficiency dropped for 1%, was slightly overpredicted (see figure 3).

4. Conclusions

In this paper the numerical predictions of the turbulent cavitating flow around a marine model propeller and a Kaplan model turbine were presented. The numerical predictions were carried out using both a commercial and an open source CFD code, solving the governing equations of the homogeneous (mixture) model.

In the case of a marine propeller three different calibrated mass transfer models were compared. The numerical results agreed well with the available experimental data and the three different mass transfer models ensured similar results, thus proving the importance of proper calibration of the models.

In the case of the Kaplan turbine the steady-state and time-dependent approaches were compared for the evaluation of the effect of the cavitation on turbine efficiency. The time dependent simulations, with the SAS SST turbulence model, better predicted the amount of cavitation and efficiency level, even though a slightly premature drop of the efficiency was observed.

5. Acknowledgements

6. References

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

[2]   Singhal A K, Athavale M M, Li H and Jiang Y 2002 Mathematical Basis and Validation of the Full Cavitation Model *Journal of Fluids Engineering* **124**(3), 617-624

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

[4]   Morgut M, Nobile E and Biluš I 2011 Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil *Int. Journal of Multiphase Flow* **37**(6), 620-626

[5]   www.openfoam.com

[6]   Jošt D, Škerlavaj A and Lipej A 2014 Improvement of Efficiency Prediction for a Kaplan Turbine with Advanced Turbulence Models, *Journal of Mechanical Engineering* **60**(2), 124-134

[7]   Salvatore F, Streckwall H and van Terwisga T. 2009 Propeller Cavitation Modelling by CFD-Results from the VIRTUE 2008 Rome Workshop *Proc. of the First International Symposium on Marine Propulsors, smp'09, Trondheim, Norway*