CAVITATION PREDICTION IN A KAPLAN TURBINE USING STANDARD AND OPTIMIZED MODEL PARAMETERS

Dragica JOŠT

University of Trieste, Department of Engineering and Architecture, Italy; currently on leave from Kolektor Turboinštitut, Slovenia

Mitja MORGUT

Kolektor Turboinštitut, Slovenia; currently on leave from University of Trieste, Italy

Aljaž ŠKERLAVAJ

University of Trieste, Department of Engineering and Architecture, Italy; currently on leave from Kolektor Turboinštitut, Slovenia

Enrico NOBILE

University of Trieste, Department of Engineering and Architecture, Italy

ABSTRACT

The paper presents efficiency and cavitation prediction in a 6-blade Kaplan turbine. The study is a result of a collaboration between the University of Trieste (Italy) and Kolektor Turboinštitut (Slovenia), which recently joined in the ACCUSIM EU project with the aim to develop reliable, high fidelity methods for accurate predictions and optimization of the performances of hydro-machinery and marine propellers.

Numerical simulations were done at one operating point for maximal runner blade angle and nominal head. Steady state results obtained with the SST (Shear Stress Transport) turbulence model were improved by transient simulations, where the SAS (Scale Adaptive Simulation) SST model was used. Cavitating flow was simulated using the homogeneous model. Mass transfer rate due to cavitation was regulated by the Zwart et al. model with default model constants used in ANSYS CFX commercial code and also with the evaporation and condensation parameters previously calibrated considering the sheet cavity flow around a hydrofoil. For a Kaplan turbine the numerical results were compared with the observation of cavity size on the test rig and with the measured sigma break curve. Steady state simulations predicted a significant too small efficiency level and too small extent of cavitation on the runner blades. With transient simulations, the shape and size of the predicted sheet cavitation agreed well with the cavitation observed on the test rig. In addition, also the predicted efficiency was more accurate, although the value of σ (cavitation or Thoma number) where the efficiency dropped for 1% was a bit too large. The difference between the results obtained with standard and calibrated model parameters of the Zwart mass transfer model was small.

KEYWORDS

Kaplan turbine, two-phase flow, cavitation, sigma break curve, Zwart mass transfer model

* *Dragica Jošt, University of Trieste, currently on leave from Kolektor Turboinštitut,Slovenia, dragica.jost@kolektor.com*

1. INTRODUCTION

In recent years an intensive research about numerical flow analysis in axial turbines was performed in Turboinštitut. The influence of modelling or neglecting hub and tip clearance was tested. Steady state simulations were done by different turbulence models on grids with different refinements near the walls. At some operating points also transient simulations were done using SST and SSG RSM models. On the basis of the results, it was concluded that none of the tested turbulence models, within steady state simulations, predicted the efficiency at all operating regimes accurately [1]. With grid refinement, some improvements were achieved but the results were still not satisfactory. Even the results of time dependent simulations with RANS models were not accurate enough. This was the motivation for time dependent simulations by more advanced turbulence models, the SAS SST and zonal LES. The improvement of results for a Kaplan turbine was presented in [2]. Even greater improvement was achieved in case of a bulb turbine [3]. Recently, in the frame of ACCUSIM project, the results for a bulb turbine were additionally improved by zonal LES model on refined grids [4] and a study of cavitation prediction with more advanced turbulence and mass transfer models with standard and optimized parameters started. Preliminary results for a Kaplan turbine were briefly outlined in [5], while a more complete selection of results is presented in this paper.

2. MATHEMATICAL MODEL

2.1. Flow equations for multiphase flow

In the homogeneous multiphase transport equation-based model, employed in this study, the cavitating flow can be described by the following set of governing equations:

$$\begin{cases} \nabla \cdot \mathbf{U} = \dot{m}\left(\dfrac{1}{\rho_l} - \dfrac{1}{\rho_v}\right) \\ \dfrac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U}\mathbf{U}) = -\nabla P - \nabla \cdot \boldsymbol{\tau} + S_M \\ \dfrac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \mathbf{U}) = \dfrac{\dot{m}}{\rho_l} \end{cases} \tag{1}$$

Cavitating flow is modelled as a mixture of two species i.e. vapour and liquid behaving as a single one. The phases are considered incompressible and share the same instantaneous velocity **U** and pressure fields *P*. The above equations are, in order, the continuity and the momentum equation for the liquid-vapour mixture, and the volume fraction equation for the liquid phase. $\boldsymbol{\tau}$ is the stress tensor, S_M stays for the additional sources of momentum, $\dot{m}$ is the interphase mass transfer rate due to cavitation, ρ_v the vapour density and ρ_l the liquid density.
The liquid volume fraction γ and the vapour volume fraction α are defined as follows:

$$\gamma = \frac{\text{volume liquid}}{\text{total volume}} \quad \alpha = \frac{\text{volume vapour}}{\text{total volume}} \tag{2}$$

and they are related to each other through the following relevant constitutive relationship:

$$\gamma + \alpha = 1 \tag{3}$$

Finally, ρ and μ are the density and the dynamic viscosity of the vapour-water mixture, scaled by the water volume fraction, respectively.

$$\begin{aligned} \rho &= \gamma \rho_l + (1-\gamma)\rho_v \\ \mu &= \gamma \mu_l + (1-\gamma)\mu_v \end{aligned} \tag{4}$$

The specific interphase mass transfer rate $\dot{m}$ can be modelled using an appropriate mass transfer model, also called cavitation model.

2.2. Turbulence modelling

In order to model the turbulent flows the governing equations have to be time-averaged leading to the well known RANS equations, widely applied to industrial flow problems. In this work the most popular two-equation eddy viscosity based models, namely k-ε and SST (Shear Stress Transport) were used for NACA hydrofoil and Kaplan turbine, respectively. An alternative approach to standard URANS (Unsteady RANS), namely the SAS (Scale Adaptive Simulation) model [6] was employed for more accurate time dependent predictions of flow in a Kaplan turbine. Due to positive experience in previous studies [1] the curvature correction (CC) and Kato-Launder limiter of production term were included in all turbine simulations. For discretization of the advection term high resolution scheme (HRS) implemented in the ANSYS CFX code was used. In [3] and [4] positive effect of bounded central differential scheme (BCDS) (in comparison to the HRS) on accuracy of torque on the shaft and efficiency prediction was clearly seen. Unfortunately BCDS in combination with cavitation modelling was found less stable and some simulations with small σ values were stopped by overflow. Therefore here only the results obtained with HRS are presented.

2.3. Mass transfer model

In this study the native mass transfer employed in CFX i.e. the Zwart et al. model was used. It is based on the simplified Rayleigh-Plesset equation for bubble dynamics.

$$\dot{m} = \begin{cases} -\mathrm{F_e}\dfrac{3r_{nuc}(1-\alpha)\rho_v}{R_B}\sqrt{\dfrac{2}{3}\dfrac{P_v - P}{P_l}} & \text{if } P < P_v \\ \mathrm{F_c}\dfrac{3\alpha\rho_v}{R_B}\sqrt{\dfrac{2}{3}\dfrac{P - P_v}{P_l}} & \text{if } P > P_v \end{cases} \tag{5}$$

In the above equations, P_v is the vapour pressure, r_{nuc} is the nucleation site volume fraction, R_B the radius of a nucleation site, F_e and F_c are two empirical calibration coefficients for the evaporation and condensation processes, respectively. The above coefficients, according to default CFX setting are equal to r_{nuc}=5×10^{-4}, R_B=2×10^{-6} m, F_e=50, F_c=0.01. In the case of the setup suggested in [7, 8] new values are F_e=300, F_c=0.03.

3. CALIBRATION OF EVAPORATION AND CONDENSATION PARAMETERS ON A HYDROFOIL

A calibration of empirical constants for three mass transfer models (the Zwart model, the FCM model and the Kunz model) using an optimization strategy is presented in [7]. The entire calibration process was driven by the modeFRONTIER 4.2 optimization system, which is a general integration and multiobjective optimization platform commonly used for functional and shape optimization of systems and devices. With the aim to reduce computational costs the empirical coefficients were optimized on a two-dimensional sheet cavity flow around the NACA66(MOD) hydrofoil. The evaporation and condensation coefficients of the Zwart model were tuned within the following ranges: 30<=F_e<=500 and 0.0005<=F_c<=0.08. The best result was found with F_e = 300 and F_c= 0.03. Details about optimization process can be found in [7].

In this section some selected results for NACA66(MOD) hydrofoil are briefly presented in order to show the differences due to usage of calibrated vaporization and condensation coefficients instead of the standard ones. In Fig. 1 (left) the suction sides pressure distributions, obtained using the native and calibrated Zwart model, are shown. It is possible to note that using the default setup the cavities corresponding to the three different cavitating flow regimes (σ=1.0, 0.91, 0.84) were underestimated. In the case of the calibrated model the results compared better with the experimental data.
In Fig. 1 (right) the cavity bubbles obtained for σ=0.91 are shown. In case of the default setup of the Zwart mass transfer model the bubble had lower vapour content then that obtained with the calibrated model. This explains the discrepancies with measurements observed for suction side pressure distributions.

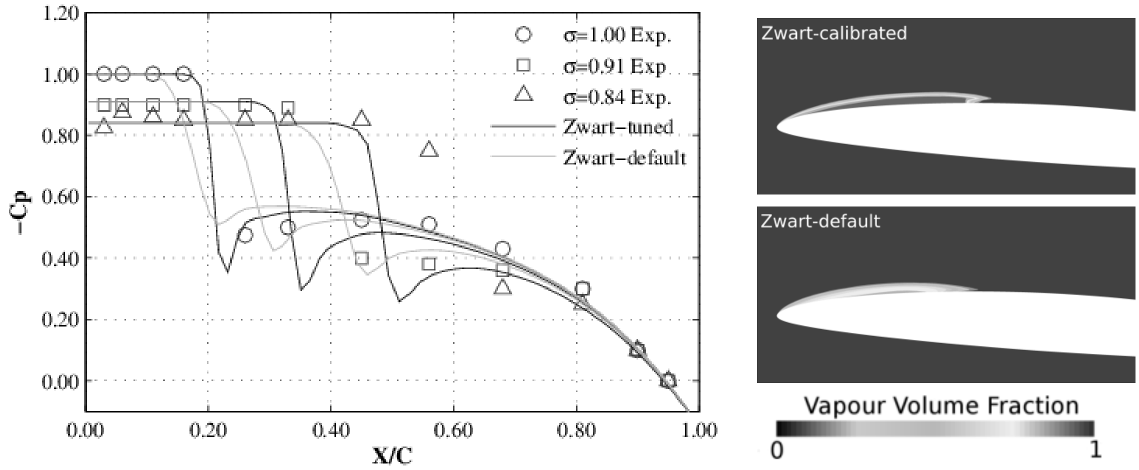

Fig 1 NACA66MOD at 4° of incidence. Suction side pressure distributions (left). Cavitation bubbles for σ=0.91 (right).

The optimized empirical constants F_e and F_c were successfully used for prediction of cavitating flow around two model scale propellers in uniform inflow [8]. Therefore we expected that the same constants will be suitable also for cavitation prediction in a Kaplan turbine.

4. CAVITATION PREDICTION FOR A KAPLAN TURBINE

Cavitation prediction was done for the same Kaplan turbine as in case of efficiency prediction presented in [2]. The turbine consists of semi-spiral casing with two vertical piers, 11 stay vanes and a nose, 28 guide vanes, a 6-blade runner and elbow draft tube with two vertical piers. All simulations were done at the operating point with blade angle 28°, flow rate coefficient φ/φ_{BEP} = 1.33, and energy coefficient ψ/ψ_{BEP} = 0.86. This operating point is close to the local best efficiency point, but its guide vane opening and flow rate are a bit larger.
The tendency for a flow to cavitate is characterized by the cavitation coefficient (or Thoma number), defined as

$$\sigma = \frac{H_a - H_s - H_v}{H} \qquad (1)$$

H_a and H_v correspond to atmospheric pressure and vaporization pressure, respectively. H is turbine head. Suction head H_s is a difference between runner blade pivot and tail water levels. The effect of cavitation on efficiency can be presented with sigma break curve which is obtained with measurements on test rig or with numerical simulations for different values of suction pressure. The aim of this paper is to numerically predict and experimentally validate a sigma break curve.

4.1. Computational domain and boundary conditions

The grid in the spiral casing with stay vanes was unstructured, while the grids in the other turbine parts were structured. Near the walls the grids were refined to get recommended values of y^+. In [2], for the draft tube and the draft tube prolongation, two grids, a basic and a refined one, were used, but it was found out that the effect of grid refinement on the results obtained with SAS SST was negligible. Therefore simulations with cavitation modelling were performed only on the basic grid with about 8.3 million nodes. The grid in the runner, which is especially important for cavitation prediction, consisted of 1.85 million nodes. Maximal value of y^+ in the runner was less than 18 while averaged value was equal 4.3. Tip clearance was modelled while hub clearance was neglected. Complete grid and a detail of grid in the runner can be seen in Fig. 2.

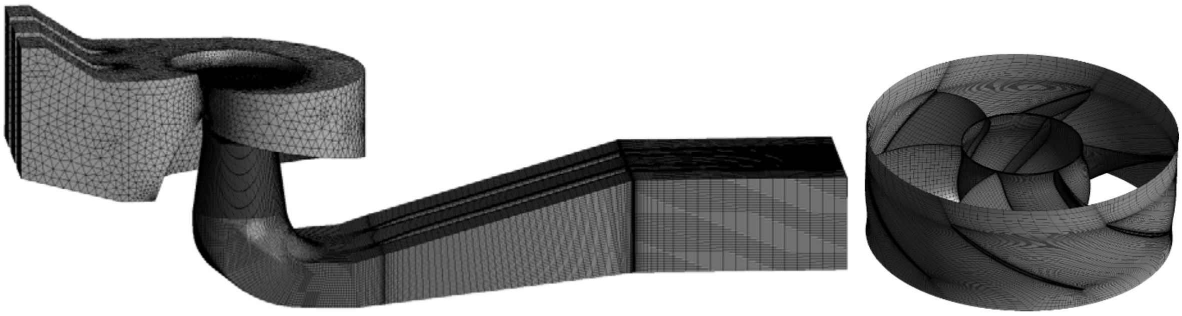

Fig 2 Computational domain and grid for Kaplan turbine (left), detail of grid in the runner(right).

Numerical simulations were performed with constant flow rate prescribed at the turbine inlet. Although the effect of gravity for the model size is small, gravity was included in computation. Therefore, a value of static pressure prescribed at the outlet of computational domain included also hydrostatic pressure. During steady-state simulations the position of runner blades relative to the stationary parts was fixed (frozen rotor condition) while during time dependent simulations the position of runner blades rotated (transient rotor stator).

4.2. Results for Kaplan turbine

To get sigma break curve several simulations with different reference pressure prescribed at the outlet of the computational domain were performed. Numerical simulations were done with prescribed flow rate, while head was a result of simulations. Contrary to the measurements, where head was almost constant for all sigma values, numerically obtained values strongly depended on accuracy of flow simulation in all turbine parts and varied due to different outlet conditions (pressure values) and also due to numerical setup (steady-state or time dependent simulation, standard or tuned mass transfer parameters).
In Fig. 3 extent of cavitation for four sigma values obtained by steady-state and time dependent simulations with standard and tuned mass transfer model parameters is presented. Sigma values obtained from the same reference pressure at the outlet of computational domain but different numerical setup are not exactly the same because sigma depends also on head which is a result of simulations. Results of simulations showed that cavitation started at tip clearance. At even smaller sigma values vapour bubbles start to appear also near the hub and finally on suction side of the blades. Comparing steady-state and time dependent results it can be seen that in case of the former ones flow started to cavitate at smaller sigma values. For all sigma values steady-state simulations predicted smaller regions with vapour bubbles.

This can be explained with larger flow energy losses in the draft tube obtained with steady-state simulations. Due to higher losses and the same reference pressure at the draft tube outlet as for time dependent simulations, pressure in the runner is higher. Besides, steady-state simulations did not predict the same extent of cavitation on all blades due to the frozen rotor condition, which somehow preserved differences in circumferential direction. With transient simulations the same amount of cavitation on all runner blades was obtained. Differences due to standard and tuned evaporation and condensation coefficients are more significant in case of steady-state simulations where with tuned parameters a bit smaller but at the same time thicker regions with vapour bubbles were obtained. For transient simulations the effect of different evaporation and condensation coefficients is less significant, with the higher influence at intermediate values of the cavitation number, i.e. $\sigma \approx 0.55$.

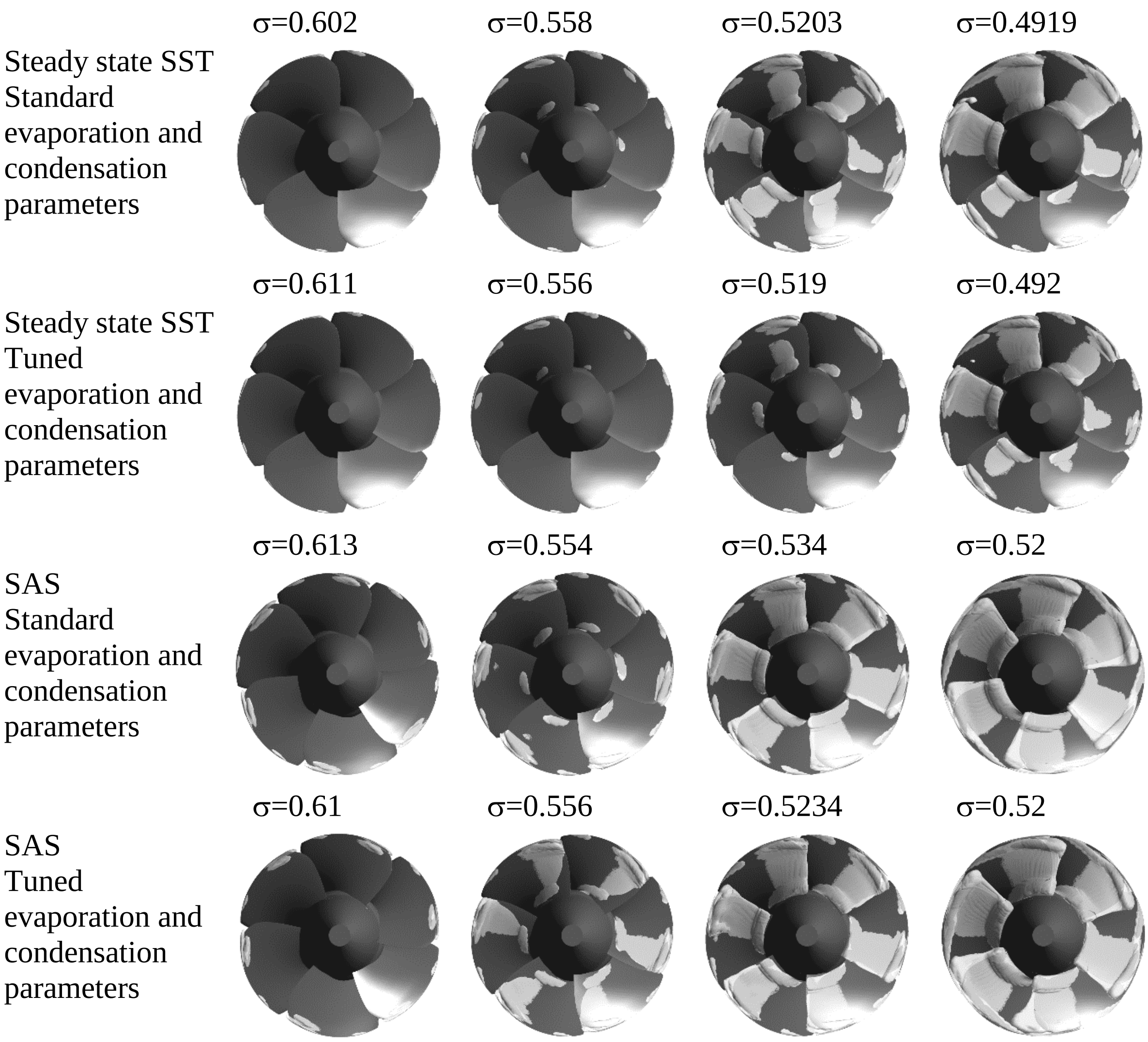

Fig. 3 Regions of cavitation, numerical results obtained with steady state and time dependent simulations for four sigma values.

In Fig. 4 cavity size at sigma value around 0.52 obtained with tuned evaporation and condensation coefficients is compared with the cavity observed on the test rig. Steady state simulations predicted too small amount of vapour bubbles while with the transient simulations predicted shape and extent of sheet cavitation agreed well with the cavitation observed on the test rig.

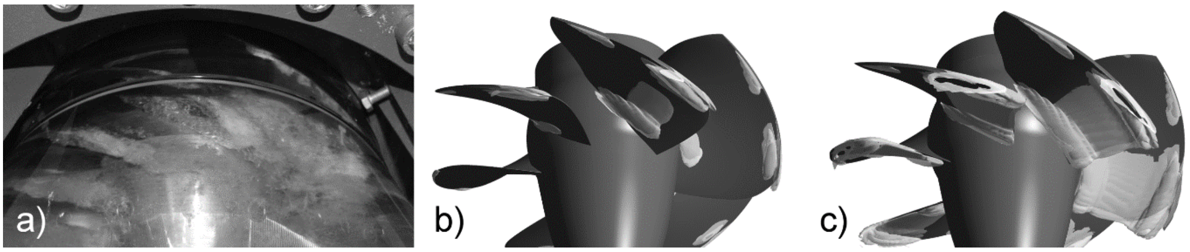

Fig.4 Comparison of shape and size of cavity, a) experiment, b) steady state simulation, tuned coefficients, c) time dependent simulation, tuned coefficients.

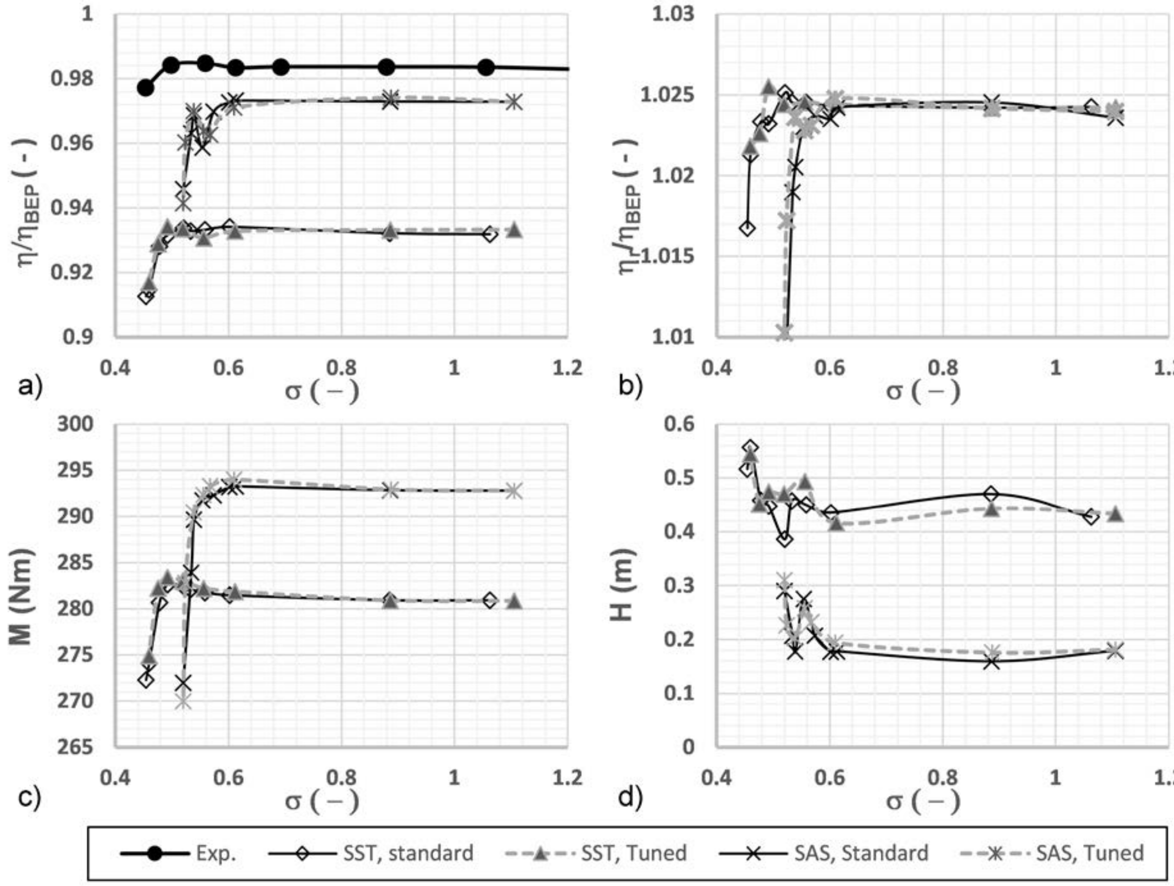

Fig 5 (a) Sigma break curve, (b) runner efficiency, (c) torque on the shaft and (d) losses in the draft tube.

In Fig. 5 besides sigma break curve also runner efficiency, torque on the shaft and flow energy losses in the draft tube are presented. Steady state simulations predicted significantly too small efficiency, but the value of sigma, where the efficiency dropped for 1%, agreed well with experimental result. Transient simulations predicted the efficiency more accurately and the whole sigma break curve moved to higher efficiency values and therefore closer to the measured curve, but a slightly premature break down of the efficiency was predicted. Level of runner efficiency for non-cavitating conditions is the same for all simulations. As already mentioned, in case of steady-state simulations due to larger losses in the draft tube cavitation started at smaller sigma values, and consequently runner efficiency and torque on the shaft break down at smaller sigma values. It has to be emphasized that part of the discrepancy between numerical and experimental results can be a consequence of different values of head and flow rate. The measurements were done with approximately constant head and some variations of flow rate, while numerical simulations were performed at constant flow rate. The head was a numerical result and varied, especially at strong cavitation. At minimum sigma value a difference between flow rate for numerical analysis and experimental value is more than 1.6%.

5. CONCLUSIONS

Steady-state simulations predicted too small extent of cavitation in the Kaplan turbine. The reason is too high pressure level in the runner due to overestimated losses in the draft tube. With time dependent simulation the same amount of cavitation on all runner blades was obtained and the shape and size of cavity agreed well with the cavitation observed on the test rig. The effect of cavitation on the machine efficiency is well reproduced by steady-state and time dependent simulations while the efficiency level is well captured only by time dependent simulations. In case of time dependent simulations a slightly premature drop of the efficiency was predicted.
Cavitation was modelled with standard and calibrated evaporation and condensation parameter for Zwart mass transfer model. With both set of parameters similar agreement with experimental results was obtained, with minimum differences of the results obtained with either the standard or calibrated model parameters.

6. ACKNOWLEDGEMENTS

The research leading to these results has received funding from the People Programme (Marie Curie Actions) of the European Union's Seventh Framework Programme FP7/2007-2013/ under REA grant agreement n°612279 and from the Slovenian Research Agency ARRS - Contract No. 1000-15-0263.

7. REFERENCES

[1] Jošt, D., Drešar, P.: *Numerical Analysis of the flow in an axial turbine by different turbulence models*. (Ljubljana: Turboinštitut) Internal report Nr. 3046 (in Slovene). 2011. pp. 49.
[2] Jošt, D., Škerlavaj, A. and Lipej, A.: Improvement of Efficiency Prediction for a Kaplan Turbine with Advanced Turbulence Models. *Journal of Mechanical Engineering*. Vol. 60. No. 2, 2014. pp. 124-134.
[3] Jošt, D., Škerlavaj, A.: Efficiency prediction for a low head bulb turbine with SAS SST and zonal LES turbulence models. *IOP Conf. Ser.: Earth Environ. Sci*. Vol. 22. No. 2. 2014. pp. 022007.
[4] Jošt, D., Škerlavaj, A., Morgut, M., Nobile, E.: Effects of turbulence model and grid resolution on the performance prediction of a bulb turbine. *NAFEMS World Congress 2015*, San Diego, USA. 2015.
[5] Morgut, M., Jošt, D., Nobile, E., Škerlavaj, A.: Numerical investigation of the flow in axial water turbines and marine propellers with scale-resolving simulations. *33rd UIT Heat Transfer Conference*. L'Aquila, Italy. 2015.
[6] Egorov, Y. and Menter, F.: Development and application of SST-SAS turbulence model in the DESIDER project. *Advances in Hybrid RANS-LES Modelling* (eds. S-H Peng and W Haase). Heidelberg: Springer. 2008. pp 261-270.
[7] Morgut, M., Nobile, E. and Biluš, I. Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil. *Int. Journal of Multiphase Flow*. Vol. 37. No. 6. 2010. pp. 620-626.
[8] M. Morgut, E. Nobile: Numerical Predictions of Cavitating Flow Around Model Scale Propellers by CFD and Advanced Model Calibration. *International Journal of Rotating Machinery*. Vol. 2012. 2012. Article ID 618180. 11 p.