Analysis of a propeller turbine operated in a full load operating point

Bernd Junginger¹, Stefan Riedelbauch¹

¹Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70550 Stuttgart, Germany
E-mail: bernd.junginger@ihs.uni-stuttgart.de

Abstract. A full load operating point of a propeller turbine is investigated both numerically and experimentally. Steady state and unsteady flow simulations, using four different turbulence models with grid sizes up to 100 million elements, are validated with experimental results which are performed according to IEC 60193 standard. Due to the high swirl number of \( S = -0.175 \) at the runner outlet, steady state and unsteady RANS simulations are no longer capable to predict the correct flow field in the draft tube of the turbine. Hence, unsteady simulations with scale-resolving turbulence models, are performed. As a result of the high swirl a full load vortex rope is generated starting at the runner hub developing in the center of the draft tube. The integral quantities head and torque as well as the axial and tangential velocity components at five evaluation lines in the draft tube are validated with experimental results. Velocity components are both time- and phase-averaged, using the runner as trigger signal. Additionally, a comparison of local quantities, head loss in the draft tube and the pressure recovery factor is carried out. The turbulence quantities of the different approaches are evaluated at the evaluation lines applying the viscosity ratio or energy spectra at discrete points in the draft tube of the turbine.

1. Introduction
A full load operating point of a low head propeller turbine is investigated. The operating point possesses the same head \( H \) and an increased discharge \( Q \) as the best efficiency point (BEP). In normalized magnitudes the operating point can be characterized with a specific speed factor of \( n_{ed} = n_{ed_{BEP}} \), a discharge factor of \( Q_{ed} = 1.12 Q_{ed_{BEP}} \) and a guide vane opening of \( \Delta \gamma = 1.36 \Delta \gamma_{BEP} \).

2. Numerical method
All simulations are carried out with the commercial CFD (Computational Fluid Dynamics) code Ansys CFX [1]. In total 4 turbulence models are applied in this study, one RANS and three hybrid RANS-LES (Reynolds Averaged Navier Stokes - Large Eddy Simulation) models. RANS models are typically used for the design process of hydraulic turbomachines. A common one is the k-\( \omega \)-SST (Shear Stress Transport) model, which is part of this study [2]. The three applied hybrid RANS-LES models, are the SAS-SST (Scale Adaptive Simulation) [3], the SBES (Stress Blended Eddy Simulation) [3, 4] and the SDES (Shielded Detached Eddy Simulation) model [3]. All applied hybrid RANS-LES models switch to a SST turbulence model for the RANS part. In the SAS model an additional source term, \( Q_{SAS} \), is introduced in the transport equation of the turbulent eddy frequency \( \omega \) to enable the turbulence model to operate in SRS (Scale Resolving Simulation) mode, and hence a LES like behavior [5]. The SBES model blends between a RANS SST and WALES model. A not yet published blending function is used to switch between RANS and LES model in both SBES and SDES. The wall protection, which is a weakness of the
DES or DDES model is hence improved [6, 7]. Both the SBES and the SDES model possess the ability of a faster transition from RANS to LES and also a clearly distinguish between RANS and LES compared to the SAS model.

A second order Euler backward scheme is used for the temporal discretization for all approaches. The spatial discretization depends on the numerical approach, while the RANS model uses a high resolution scheme, a central differencing scheme is applied for all hybrid turbulence models [8, 9]. A first order scheme is applied for the turbulent quantities in spatial discretization [3].

3. Numerical and experimental setup

The model sized propeller turbine is installed in the closed test loop at the laboratory of the Institute of Fluid Mechanics and Hydraulic Machinery. The validation is performed according IEC 60193, which defines the standard for model acceptance test of hydraulic machines [10].

A CAD-model of the hydraulic contour is illustrated in figure 1. The integral validation quantities are the turbine head and the runner torque. The turbine head, according to IEC 60193 standard, is measured by a differential pressure gauge and the kinetic energy at control cross-section. The axial and tangential velocity components are measured at discrete points on evaluation lines M1-M5 with an 2D-LDA system. The evaluation lines are positioned D/2, D, 2D, 2.6D and 4D downstream the runner. An inductive proximity switch at the runner shaft serves as trigger signal, so all data can be both time- and phase-averaged. Time-resolved wall pressure signals are recorded at the control cross section DP1, DP2 and DP3 which is about D/2, D, 2D downstream the runner.

In total five different numerical grid densities are investigated. A steady state approach with 5M (million) elements and unsteady approaches with 15M, 30M, 50M and 100M are analyzed. The RANS SST model is applied for a mesh density of 30M. Hybrid RANS-LES turbulence models are applied to the 100M grid density. The influence of the computational grid is evaluated by a grid study using the SBES turbulence model. A study on the influence of the runner gap is performed with a mesh density of 50M and the SBES turbulence model. The normalized runner gap size \( \tau \) is defined as \( \tau = \frac{s_{gap}}{s_{gap_{max}}} \) with the actual gap size \( s_{gap} \) and the maximal investigated gap size \( s_{gap_{max}} \). The investigated gap sizes in the study varies between 0 - 1.0. In the experiment the runner gap size is \( \tau = 0.3 \). A velocity profile and turbulence quantities are applied at the inlet of the computational domain for all simulations. A static pressure is set at the outlet of a storage tank, which is placed downstream the draft tube, to decouple the boundary condition. Depending on the mesh size the time step is set so the CFL number is smaller then 1. The dimensionless wall distance \( y^+ \) is 1 or smaller for all grid densities except for 5M and 15M which possess a \( y^+ \)-value smaller then 10.

4. Results

Both experimental and numerical results are time- and phase-averaged. However, the amount of data differs. The integral quantities of the experimental results are time averaged over 30s. The LDA system records 20 000 bins for each measuring point. The numerical approaches are time averaged over 50 runner revolutions. The number of available time steps is depending on the applied time step. In both cases, experiment and simulation, the averaging process starts when a periodic flow behavior is achieved.
3. \( \tau = 0.3 \).

4.1. Global machine data

The integral quantities head and torque of the different numerical approaches are validated with the experimental results. The results are illustrated in figure 2. The steady state approach is not capable to predict both head and torque within the measurement precision. A comparison of the results with the nominal gap size of \( \tau = 0.3 \) shows that only the SBES turbulence model is able to correctly predict, both head and torque. All unsteady simulations are basically able to predict the correct runner torque independent of the turbulence model. The critical quantity is the turbine head which is a result of the losses of the different machine components. Essential for good simulation results is the correct computation of the flow field in the draft tube.

4.2. Flow analysis

Time-averaged velocity of the axial and circumferential components are compared with results obtained by the 2D-LDA system, illustrated in figure 4. The stagnation region in the center of the draft tube, downstream the runner hub predicted by the RANS model is too small compared to the experimental results. The coarse grid density of 15M with the SBES turbulence model also shows small deviations to the validation values. Further downstream the runner at evaluation lines M4 and M5 also the SAS approach applied on a 100M grid shows deviations compared to the experimental results. Both approaches, SBES and SDES, on mesh densities with 50M or more elements show good agreement with the LDA results.

The effect of the runner gap on the flow field can mostly be observed at evaluation lines M1 and M2. Due to periodicity reasons a segment of one runner blade, which corresponds to 90°, can be used for the comparison. The phase averaged axial and circumferential velocity contour of M1 for different runner gap sizes \( \tau = 0.3 \) is presented in figure 5.
Figure 4: Time-averaged axial and circumferential velocity profiles at the evaluation lines M\textsubscript{1} - M\textsubscript{5} for a runner gap size of \(\tau = 0.3\).  

Figure 5: Phase-averaged axial and circumferential velocity contour of simulation results using a 50M mesh density and the SBES turbulence model at evaluation line M\textsubscript{1}. The tip of the runner blade can clearly be identified in the phase-averaged plots at evaluation line M\textsubscript{1} in both axial and circumferential velocity component. A larger runner gap \(\tau\) leads to a stronger tip vortex close to shroud. For a runner gap of \(\tau = 0.5\) and \(\tau = 1.0\) the tip vortex splits up in two or more vortex streaks. Due to a nonphysical boundary condition a vortex close to the shroud develops even for the simulation approach without runner gap. The shear strain between the rotating runner and the standing shroud leads to the development of a vortex similar to the gap vortex. A larger runner gap also leads to the development of a stronger vortex starting from the runner hub this can be observed in the reduction of the velocity components. Hybrid RANS-LES models and a rather fine grid makes it possible to see the guide vanes in the phase-averaged flow field at evaluation line M\textsubscript{1} which are not resolved by the RANS simulation.  

The pressure gradient between the pressure and the suction side causes a jet with higher velocity components close to the shroud, shown in figure 6. With an increasing runner gap size the axial velocity close to the shroud along the draft tube wall is on a higher level. This
results in an axial extended stagnation region in the center of the draft tube downstream the runner rotation axis. The higher axial velocity component close to the hub in combination with the remaining circumferential velocity downstream the runner leads to a better draft tube performance, which can also be seen in figure 3.

4.3. Turbulence quantities

Depending on the numerical approach the ability to resolve turbulent flow structures is evaluated by the viscosity ratio, illustrated in figure 7. The viscosity ratio of the RANS model is about one order of magnitude higher than SAS turbulence model with a grid density of 100M elements. SBES and SDES turbulence models are almost an additional magnitude lower in the viscosity ratio. At evaluation line M1 the full load vortex rope developing in the center of the draft tube causes an increase of the eddy viscosity ratio noted by all numerical approaches.

Another option to evaluate the performance of turbulence models is a turbulent energy spectra, presented in figure 7 for two discrete points. The first point is located on evaluation line M1 in the draft tube cone, and the second on line M5 close to the draft tube outlet. While at line M1 the large scales like the runner frequency can still be seen in the energy spectra only the decay of isotropic turbulence is visible at line M5. A significant difference between the turbulence models can be noted in the ability to resolve the initial scale of -$5/3$. The RANS model is not able to resolve the initial scale at all and the SAS model has considerable limitations compared to the SBES and the SDES turbulence model.

Figure 6: Time-averaged axial velocity (top) and standard deviation of the axial velocity (bottom) for different runner gap sizes with a 50M mesh and the SBES turbulence model plotted on n cut plane in the middle of the draft tube

Figure 7: Eddy viscosity ratio at evaluation lines M1 (top left) and M5 (top right) for all simulation approaches with a runner gap size $\tau = 0.3$. Turbulent kinetic energy at evaluation lines M1 (bottom left) and M5 (bottom right) for the investigated turbulence models a 30M and 100M grid with a gap size of $\tau = 0.3$
4.4. Dynamic pressure signals
Figure 1 shows the three positions in the draft tube cone (DP1 - DP3) at which the pressure signal is recorded. The runner gap size has an influence on the amplitude of the pressure signal which can be analyzed by performing a FFT (Fast Fourier Transformation) of time domain signal. A detail of the signal in frequency domain is displayed in figure 8. The characteristic amplitude at four times the runner frequency (four runner blades) increases with an increase of the runner gap size $\tau$. This phenomena can be observed at all three measuring positions in the draft tube cone. However, the gap size effect is quickly reduced in downstream direction.

5. Summary and Conclusion
Various sets of parameters (turbulence models, mesh size, tip clearance size) are used to investigate a full load operating point of a low head propeller turbine. A validation with experimental results is presented for integral quantities head and torque as well as for time-averaged velocity profiles at 5 evaluation lines in the draft tube. Moreover, the effect of the runner gap size on the flow field is investigated. The RANS approach lacks in result of the quality for the integral quantities as well as for the velocity components. Compared to newer implemented hybrid RANS-LES turbulence models the SAS models show some weaknesses in predicting the turbine head losses and flow separation in the draft tube diffusor. A larger runner gap size increases the draft tube performance. However, the reduction of the runner torque and hence the turbine efficiency cannot be counterbalanced by the reduction of the draft tube head losses. Larger runner gap sizes lead to stronger tip vortices and with gap sizes larger than $\tau = 0.5$ to multiple vortex streaks. The runner gap size can also be seen in the amplitude of wall pressure signals in the draft tube cone. Pressure amplitudes increase with increasing gap size.

Acknowledgment
The authors gratefully acknowledge the High Performance Computing Center Stuttgart for providing computational resources.

References
[1] ANSYS, Inc. Canonsburg, PA, USA 2017 ANSYS CFX-Solver Theory Guide Release 18.0
[2] Menter F R 1994 AIAA-Journal 32 1598–1605
[3] Menter F R 2015 Best Practice: Scale-Resolving Simulations in ANSYS CFD Version 2.0
[4] Menter F R 2016 6th Symposium on Hybrid RANS-LES Methods
[5] Menter F R, Schütze J and Gritskievich M 2012 Progress in Hybrid RANS-LES Modelling (Springer) pp 15–28
[6] Strelets M 2001 39th Aerospace Sciences Meeting and Exhibit (Reno, NV, USA) p 879
[7] Shur M L, Spalart P R, Strelets M K and Travin A K 2008 International Journal of Heat and Fluid Flow 29 1638–1649
[8] Barth T J and Jesperson D C 1989 27th AerospaceSciences Meeting (Reno, NV, USA)
[9] Jasak H, Weller H and Gosman A 1999 International journal for numerical methods in fluids 31 431–449
[10] International Electrical Commission (Editor) 1999 International standard IEC 60193 Second Edition 1999-11. Hydraulic turbines, storage pumps and pump-turbines - Model Acceptance Tests
[11] Junginger B and Riedelbauch S 2018 High Performance Computing in Science and Engineering'17 (Springer) pp 355–368
[12] Junginger B and Riedelbauch S 2018 20th International Seminar on Hydropower Plants - ViennaHydro