A Turbulence Model Assessment for Deep Part Load Conditions of a Francis Turbine

J Wack\textsuperscript{1}, J Beck\textsuperscript{2}, P Conrad\textsuperscript{2}, F von Locquenghien\textsuperscript{2}, R Jester-Zürker\textsuperscript{2} and S Riedelbauch\textsuperscript{1}

\textsuperscript{1}Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70569 Stuttgart, Germany
\textsuperscript{2}Voith Hydro Holding GmbH & Co KG, Alexanderstraße 11, 89522 Heidenheim, Germany
E-mail: jonas.wack@ihs.uni-stuttgart.de

Abstract. Today, hydropower plants received increased attention as they have the ability to stabilize the electrical grid. However, this is accompanied by more start-stop sequences as well as operation in off-design conditions. One big task is to develop improved runner designs that can resist the increased pressure fluctuations. In the scope of this study, a Francis turbine is simulated for a deep part load operating point with the use of the SAS model and the more recently developed SBES model with ANSYS CFX. Both turbulence models are hybrid RANS-LES models. The simulation results are compared against model test data from a measurement at high cavitation number. For the evaluation of the pressure fluctuations comparison is furthermore made against experimental data at low cavitation number. All in all, SAS and SBES model predict similar flow characteristics. Both, the velocity profiles in the PIV measurement plane and the predicted pressure fluctuations, show only minor differences. The comparison against experimental data shows that the amplitude of the broadband frequency spectrum, which results from the occurrence of vortices, is overestimated at the investigated locations on the runner blades.

1. Introduction

As the share of new renewable energy sources (NRE) on the energy mix increases, so does the need for other energy sources which are able to compensate the volatility of e.g. wind and solar power. Here, responsive hydropower plants play a decisive role. Today, water turbines see more start-stop sequences than ever before, operate longer on stand-by and are operated outside the designed operating range [1]. Premature component fatigue might be the consequence. Adaptations of future runner designs for enhanced part load operation require reliable load evaluations. For deep part load conditions (small guide vane opening) the flow and associated pressure fluctuations become increasingly stochastic [2] and numerical simulations using two-equation turbulence models fail to depict this behavior [3]. Consequently, for this operating range algebraic stress models [4] or hybrid RANS-LES models [3, 5] are typically used for numerical simulations. The goal of this study is to assess the ability of SAS (Scale Adaptive Simulation) and SBES (Stress-Blended Eddy Simulation) turbulence model to accurately capture flow characteristics like velocity distribution and pressure fluctuations by validating simulation results with experimental data.
2. Experimental Setup

A model test with multiple sensors in the stationary and rotating frame of a model Francis runner has been performed. This includes both pressure and strain gauge sensors on the runner blades. In total 30 sensors have been installed in the rotating system with 16 pressure and 12 strain gauges on multiple blades. The sensors are partly buried in the blade to not influence the flow. Observations did not show any influence e.g. in terms of cavitation due to the sensors. It was chosen to introduce pressure sensors on both sides of one blade to see the load on one blade. The sensors for strain gauge testing were spread across three blades at both positions crown suction side and band pressure side, as locations of expected highest dynamic stress. Additionally, pressure sensors were installed in the stationary system such as draft tube cone (DTCo) or vaneless space (VS) at IEC conform positions. In total six sensors at different angles are available for the draft tube cone position. Figure 1 shows the sensor allocation. The odd numbers on the blade sensors refer to the sensors on the suction side and the even to the pressure side. In this publication only the results of the pressure sensors are discussed. The results of the strain gauge measurements are presented in [12].

The runner design is of a mid-range specific speed \( n_q = 40 - 60 \) homologous scaled to model test machine size. The test rig utilized for the investigations is a standard test rig for hydraulic model testing. A large test program was performed with multiple operating conditions. For this investigation a deep part load operating point is chosen that has the following characteristics: discharge \( Q/Q_{BEP} \approx 0.33 \), discharge factor \( Q_{ed} = 0.07 \), speed factor \( n_{ed} = 0.37 \) and 8° guide vane opening. Figure 2 shows the characteristic peak-to-peak values of the pressure pulsation in the draft tube cone for the specific head in relation to the operating point. The operating condition is dominated by stochastic pressure fluctuations with clearly developed channel vortices (compare Figure 3). Figure 2 and 3 are in accordance with the boundary conditions during a prototype measurement. The tail water level was raised for one measurement to suppress the cavitation volume in the draft tube cone and enabling PIV measurements, while keeping \( n_{ed} \) and \( Q_{ed} \) constant. However, a cavitation volume was observed at the inter-blade vortices even for the high cavitation number. Additionally, pressure measurements were performed at low cavitation number and consequently a significantly higher cavitation volume.

2.1. PIV Measurements

In order to get insights into the flow field, Stereoscopic PIV measurements are conducted. PIV allows the simultaneous capturing of a vector field in a measuring plane with a spatial resolution of less than 3 mm in each direction for the given setup. A ND-YAG laser was used to create a light sheet at the measuring plane below the runner. The optical access was achieved via a recess in the spiral-case housing. Hollow glass spheres with a diameter of 10 µm served as tracer particles. The laser light reflections by the particles are observed by two cameras, which allows...
Figure 2. Draft tube cone pressure pulsations for different operating points. The investigated operating point is marked by the red line.

Figure 3. Channel vortices at deep part load (lower σ compared to PIV measurement to visualize the inter-blade vortices).

for conclusions about their convection with the flow and therefore enables the calculation of the flow vector. The draft-tube cone was made of PMMA to have optical access for the cameras to the measurement plane.

Figure 4. Stereo PIV setup: tracer particles, convected by the flow, get illuminated by a laser lightsheet and are recorded by two cameras.

2.2. PIV Data Processing
Due to a challenging positioning, steep observation angles, strong curvatures and total reflection, the evaluation of the measuring plane was limited, especially in the peripheral areas and therefore were masked out accordingly. Because of the challenging optical conditions, calibration is of particular importance. The common method of polynomial image correction did not rectify the distorted image sufficiently well. Therefore the pre-dewarping method (see [6] for more information) was applied. The operating point was measured in a triggered manner, one recording every second runner revolution, so that an averaged velocity field could be determined. The vector field thus reflects the flow at a specific runner position.

The measurement covers 1000 instantaneous recordings from which the temporally averaged velocity field, along with other quantities such as standard deviation and higher statistical quantities such as Reynolds stresses were derived. While these vector fields are interesting to detect locally resolved flow phenomena, such as wake flow, circumferentially averaged profiles are of further interest. For this purpose, measured data were interpolated on a polar coordinate system and a velocity profile was calculated over the radius by averaging along the circumference. Therefore velocity fluctuations along the circumference can be determined, in addition to the
velocity fluctuations over time. For comparisons with simulated results, it is therefore useful to take into account the circumferential and temporal fluctuations, as well as the measurement uncertainty as bands. It should be mentioned that due to the masking of the data at higher outer radii there are only few data points left for the circumferential averaging. The coverage of measured circumference describes the ratio of usable data on the circumference in percent.

3. Numerical Setup

Within this study, all simulations are performed with ANSYS CFX (version 19.2) for a Francis turbine at model scale, which consists of 24 guide vanes and 13 runner blades. The simulation domain ranges from the inlet of the spiral case (SC) to the outlet of the downstream tank. Single-phase simulations are performed, which means that the impact of cavitation is neglected in these investigations. To evaluate the impact of the turbulence model, one simulation is performed with the SBES model while another simulation uses the SAS model. Both models are based on hybrid RANS-LES modeling. A standard RANS approach is not considered in this study as previous work showed the insufficient suitability of capturing the stochastic nature of pressure fluctuations at deep part load conditions [3]. The selection of the SAS and SBES model is based on significant differences between these models for the investigation of a propeller turbine [7]. Both models have in common that they use SST turbulence model for the RANS part. However, the SAS model uses an additional source term in the transport equation for the turbulent eddy frequency, which results in the LES-like behavior, while the SBES model directly blends between RANS SST and WALE LES model.

For all simulations a bounded central differencing scheme is applied for spatial discretization and a second order backward Euler scheme for temporal discretization. Before the averaging process is started, 20 runner revolutions are carried out within the initialization process. The averaging process is then performed for 80 runner revolutions. Due to the application of hybrid RANS-LES turbulence models the time step needs to be set accordingly. For the present study, the time step corresponds to approximately 0.4° of runner revolution, which results in an averaged Courant number of 0.8. The time step size is selected based on the sampling rate of the experimental data for the pressure sensors. Within a time step four coefficient loops are
performed. The RMS residuals after the final coefficient loop are around $2 \cdot 10^{-4}$ for the velocity components and $2 \cdot 10^{-5}$ for the pressure.

The boundary conditions are set as follows: A constant mass flow is prescribed at the inlet of the spiral case and static pressure is set at the outlet. As runner seal leakage is almost independent of the operating point [8], it can be expected that it has more effect on the velocity profile for deep part load operating point compared to operating points with higher discharge. To include the seal leakage into the model, a sink is applied at the runner inlet using an outlet boundary condition and a source is prescribed at the runner outlet by an inlet boundary condition. The discharge through side chamber is assumed to be 0.33% of the discharge for the investigated operating point. Between neighboring stationary subdomains a general grid interface is used and a transient rotor stator interface is applied between stationary parts and the rotating runner.

To properly resolve the inter-blade vortices the use of a fine mesh is necessary [9, 10]. Furthermore, the usage of hybrid RANS-LES turbulence models requires a suitable mesh resolution. Within this study, a mesh with approximately 25 million cells is used. The cell distribution is presented in table 1. A preliminary study with steady state RANS simulations showed the highest sensitivity for head and torque for the mesh resolution of the subdomains covering stay and guide vanes (SVGV) and the runner (RU) and were therefore refined accordingly. As only one mesh size is considered in this study, the mesh sensitivity regarding vortex resolution cannot be evaluated in this study and will be part of future investigations. Furthermore, it has to be mentioned that the interface between the subdomains runner and draft tube (DT) is located at the end of the draft tube cone to ensure that the interface is far away from the inter-blade vortices.

Table 1. Mesh size of the subdomains for used grid in million elements.

| SC | SVGV | RU | DT | Tank | Total |
|----|------|----|----|------|-------|
| 0.5 | 7.9  | 10.8 | 5.2 | 0.6  | 25.0  |

4. Results

The investigated deep part load operating point is characterized by the occurrence of inter-blade vortices and stochastic pressure fluctuations. Predicted head is underestimated by 4.5% with the SBES model and 4.0% with the SAS model. Both models overestimate the torque by 7.1%.

4.1. Numerical Comparison

Viscosity ratio $\nu_t/\nu$ can be used to evaluate whether there is a high share of modelling in terms of turbulent structures. High values stand for a high share of modelling ($\nu_t$ is high), while low values correspond to a fine resolution of turbulent vortices. The averaged viscosity ratio for the SAS and SBES model as well as the shielding function of the SBES model are displayed in Figure 6 for a plane through the turbine. Over a wide range the viscosity ratio is lower for the SBES model. However, at some locations in the runner high values of viscosity ratio can be observed for this model. These locations can be found in the regions where the inter-blade vortices occur. Consequently, for the SBES model a high amount of modeling takes place at the location of the inter-blade vortices. Furthermore, it is conspicuous that for both turbulence models it comes to a sudden jump of viscosity ratio at the end of the draft tube cone. This can be explained by the higher mesh resolution of the runner subdomain, which as a consequence results in the increased viscosity ratio over the interface to the draft tube subdomain.

The shielding function $f_s$ is part of the SBES model. Its functionality is to blend between RANS ($f_s = 1$) and LES ($f_s = 0$). The analysis of the shielding function of the SBES model
shows that over a wide range LES is applied. Only in regions close to the wall and in some regions in the spiral case RANS model is used. In the region of the inter-blade vortices LES is performed. To reduce the high amount of modeling represented by the high viscosity ratio, it would be necessary to significantly refine the mesh in the region of the inter-blade vortices.

In terms of computational effort both turbulence models do not show a significant difference. Furthermore, the level of the residuals is similar.

4.2. Flow Characteristics
For the PIV measurement plane (see Figure 1) a comparison between experimental and simulation results can be made for the axial, circumferential and radial velocity component. As described in section 2.1 the experimental results are averaged from 1000 samples that come from a triggered signal of every second runner revolution. For the simulation, data is triggered on the passing of the blades. Consequently, 1040 samples are used for the averaged results (80 runner revolutions). The results for the circumferentially averaged velocity profiles are shown in Figure 7. In the middle of the draft tube cone (dimensionless span $s < 0.45$), the axial velocity indicates that for both turbulence models the backflow is slightly overestimated. Unfortunately, in the region close to the draft tube wall the accuracy of the measurement is reduced as only few valid data points are available (see Figure 5) and consequently experimental results in this region have to be taken with caution. A comparison of SAS and SBES model shows only minor differences in the axial velocity distribution.

![Figure 6. Viscosity ratio for SAS (left) and SBES (middle) model and shielding function of the SBES model (right).](image)

![Figure 7. Axial (left), circumferential (middle) and radial (right) velocity distribution for the location of the PIV measurement. Results are time and circumferential averaged.](image)
For the circumferential velocity, simulations and experiment show a good agreement in the middle of the draft tube cone ($s < 0.5$). Contrary to axial velocity, the circumferential velocity profile between SAS and SBES model differ from each other. In the region $0.6 < s < 0.8$ the results of the SBES model are in better agreement with the measurement.

Radial velocity is much smaller compared to axial and circumferential component. While close to the draft tube wall ($s > 0.8$) the results for SBES and SAS model are quasi identical, in the inner part the SBES model has a slightly higher radial velocity. Compared to the experiment, the simulation is in a similar range for the radial velocity component. The wiggles in the experimental data for $s < 0.4$ are caused by measurement uncertainties and the oscillations around $s \approx 0.8$ are probably a result of the significantly reduced amount of valid data close to the draft tube wall (compare Figure 5).

To get an impression of the flow structure in one runner channel, the vortices are visualized by an isosurface of the velocity invariant $Q$ of a random time step (see Figure 8). It can be observed that the main vortex structure is the inter-blade vortex that ranges from the hub close to the leading edge to the runner outlet, where it is located close to the shroud. However, two further characteristic regions can be detected. First, a stagnation region is present at the trailing edge of the runner close to the hub that can also be found in the measurement (see Figure 3). Furthermore, vortices are moving upstream close to the suction side of the runner blade.

4.3. Pressure Fluctuations

In Figure 9, for both simulations the averaged pressure and the corresponding standard deviation are displayed on the pressure and suction side of a runner blade. Furthermore, an isosurface of the averaged velocity invariant $Q$ is shown. Contrary to Figure 8 only the inter-blade vortex and the stagnation region show strong vortex structures for the averaged values. The reason is that the stagnation region is static and the movement of the inter-blade vortices is significantly lower compared to the strong movement of the vortices close to the suction side. The impact of the inter-blade vortex on the pressure field can especially be seen on the suction side. There, a region of lower pressure that follows the course of the inter-blade vortex can be found in the averaged pressure field. In the standard deviation this course can be seen even better. Along the path of the inter-blade vortex a high standard deviation is present, which corresponds to high pressure fluctuations.

On the pressure side the impact of the inter-blade vortex can only be found near the trailing edge towards the shroud. The reason for the higher pressure fluctuations on the suction side is that over a wide range of the blade passage the inter-blade vortex is located close to the suction side. Only just before the end of the blade passage the location of the vortex is quite in the middle between pressure and suction side. Furthermore, the vortices that move upstream have a significantly higher impact on the suction side. Especially in the region of point P7 these vortices cause a high standard deviation and consequently high pressure fluctuations. All in all, the standard deviation shows that much higher pressure fluctuations are present on the suction side. A comparison of the turbulence models shows only minor differences. While close to the leading edge the SAS model shows a higher peak of the standard deviation on the suction side, the SBES model predicts higher pressure fluctuations close to the trailing edge of pressure and suction side.
In Figure 9, the location of the points P3, P7 and P8 are marked. It can be seen that the points close to the shroud (P7 and P8) are located in a region of high pressure fluctuations. While for P7 this results from the upstream moving vortices close to the suction side, P8 is affected by the inter-blade vortex.

For the points P3, P7, P8, VS, DTC00 and DTC0180 the results of an FFT are presented in Figure 10. Both, for simulations and measurements, $2^{16}$ samples are used, which corresponds to approximately 72 runner revolutions. In total, the available experimental data covers around 600 runner revolutions. From that an arbitrary time signal is depicted to have the same frequency resolution compared to the simulations. The use of 72 runner revolutions is in agreement to findings on a different Francis turbine, where the spectra of 80 and 800 revolutions were almost identical [11]. A comparison between the three locations that are located close to the runner trailing edge shows that the amplitudes of pressure fluctuations are higher at the shroud (P7 and P8) compared to the hub (P3), which is in agreement with the results from Figure 9. The reason is that at location P3 the pressure fluctuations are low due to the influence of the stagnation region, while over a wide region of the suction side the moving vortices (e.g. at P7) and the inter-blade vortex cause significantly higher fluctuations. Furthermore, the inter-blade vortex affects the fluctuations at point P8 on the pressure side of the blade.

The FFT results show the typical stochastic behavior for a deep part load operating point that can be seen by the occurrence of many different peaks for low frequencies ($f/f_{RU} < 10$). The large number of frequencies in this range with relevant amplitude can be critical in terms of fatigue. It is conspicuous that the various peaks are only present for locations P7 and P8, while at P3 one characteristic peak around $f/f_{RU} = 1$ is present. This can most likely be explained with the influence of vortical structures (upstream moving vortices for P7 and inter-blade vortex for P8) that are not present for location P3. Basically, it can be determined that both hybrid RANS-LES turbulence models can capture the stochastic behavior of the pressure fluctuations. However, a comparison to the experimental data shows that especially for location P7 the simulation results overestimate the pressure amplitude, while a comparison between the results from SAS and SBES model indicate that both models predict the amplitude in the same order of magnitude.
There are two reasons that can explain the overestimation of the pressure fluctuations at locations P7 and P8. First, simulation inaccuracies can result in a deviation of the prediction of vortex structures. This might result in a slightly different location of the vortices between simulation and experiment. In Figure 9, it can be seen that points P7 and P8 are located in a region of high standard deviation, which corresponds to high pressure fluctuations. However, just a little closer to the shroud the standard deviation is significantly lower and consequently smaller pressure fluctuations can be expected there. At the investigated off-design operating point a reduction of simulation inaccuracies should be possible by a mesh refinement as this results in a better resolution of secondary flow phenomena. Preliminary results indicate that the overestimation of pressure fluctuations can be reduced by mesh refinement, which will be investigated more detailed in the future.

The second reason for the overestimation of the pressure fluctuations is that the occurrence of cavitation has an impact on the amplitude of the pressure fluctuations. This can be seen in the experimental data for two different cavitation numbers. For locations P3 and P8 the results with bigger cavitation volume (low $\sigma$) have a smaller amplitude. Consequently for these locations the presence of a bigger cavitation volume at the static stagnation region or the inter-blade vortices result in a damping of the pressure fluctuations. For location P7 the amplitude is in a similar range for both cavitation numbers. There, no damping caused by the cavitation volume difference can be observed, which might be a result of the strong movement of the vortices. For all locations on the runner blade it can be observed that the measurement with high cavitation number has peaks caused by rotor stator interaction ($f/f_{RU} = 24$) that are not present for the low cavitation number.

In the vaneless space (VS) the most characteristic peaks in the FFT result from the rotor stator interaction. There, the simulations underestimate the amplitude at blade passing frequency ($f/f_{RU} = 13$) and overestimate the peak at $f/f_{RU} = 26$. For the locations in the draft tube cone (DTC00 and DTC0180) the broadband frequency spectrum caused by the inter-blade vortices can be observed in experiment and simulation results. Compared to the sensors at the runner trailing edge (P3, P7 and P8) the amplitude is smaller. Again it can be observed that
the amplitude is similar for the results of SAS and SBES model regarding the broadband noise. However, for blade passing frequency the SBES model predicts a smaller amplitude compared to the SAS model. For small frequencies \((f/f_{RU} < 3)\) simulation results show a similar amplitude compared to the measurement with high cavitation number and are smaller compared to low cavitation number. For higher frequencies both simulations overestimate the amplitude of the broadband noise.

5. Conclusion and Outlook

The assessment of the simulation results of SAS and SBES model showed differences for the viscosity ratio and consequently the resolution of turbulent vortices. Surprisingly, both models predict similar velocity profiles and pressure fluctuations. Although differences in the standard deviation of the pressure on pressure and suction side of a runner blade can be observed, these differences are very local. A validation against experimental data showed that the amplitudes of the broadband frequency spectrum that is characteristic for deep part load operation are overestimated by the simulations. The comparison of the experimental data at two different cavitation numbers shows that the size of the cavitation volume has an influence on the amplitude of the pressure fluctuations.

The goal for future work is to have a better agreement with the experimental results. For that different measures will be investigated. First, a better resolution of turbulent vortices can be expected with a finer mesh. Furthermore, two-phase simulations will be carried out that can consider effects that result from the vapor phase like a changed amplitude of the pressure fluctuations. The CFD results have been used as input for FEM simulations that are described more detailed in [12]. Based on the findings of this project it should be derived how reliable load evaluations can be made with highest simulation efforts.

Acknowledgments

The authors acknowledge the financial support by the Federal Ministry for Economic Affairs and Energy of Germany in the project FrancisPLUS (project number 03EE4004A).

References

[1] Seidel U, Mende C, Hübner B, Weber W and Otto A 2014 IOP Conference Series: Earth and Environmental Science vol 22 (IOP Publishing) p 032054
[2] Dörfler P, Sick M and Coutu A 2012 Flow-induced pulsation and vibration in hydroelectric machinery: Engineer’s guidebook for planning, design and troubleshooting (Springer)
[3] Wack J, Riedelbauch S, Yamamoto K and Avellan F 2016 Proceedings of the 9th International Conference on Multiphase Flow (Florence, Italy)
[4] Conrad P, Weber W and Jung A 2017 Proceedings of the Hyperbole Conference, Porto, Portugal
[5] Yamamoto K 2017 Hydrodynamics of Francis turbine operation at deep part load condition Ph.D. thesis École Polytechnique Fédérale de Lausanne
[6] Beck J, Jossberger S, Jester-Zürker R and Riedelbauch S 2019 IOP Conference Series: Earth and Environmental Science vol 405 (IOP Publishing) p 012024
[7] Junghinger B and Riedelbauch S 2019 IOP Conference Series: Earth and Environmental Science vol 405 (IOP Publishing) p 012028
[8] Čelič D and Ondráčka H 2015 Journal of Physics: Conference Series vol 579 (IOP Publishing) p 012007
[9] Stein P, Sick M, Dörfler P, White P and Braune A 2006 Proceedings of the 25th IAHR Symposium on Hydraulic Machinery and Systems, Yokohama, Japan p 228
[10] Wack J and Riedelbauch S 2015 Journal of Physics: Conference Series 656 012074 ISSN 1742-6596
[11] Conrad P, Weber W, Maiwald M, Jung A and Loefflad J 2017 The International Journal on Hydropower and Dams (Sevilla, Spain: Hydro 2017)
[12] von Loquenghien F, Wack J, Conrad P, Jester-Zürker R and Riedelbauch S 2021 Proceedings of the 30th IAHR Symposium on Hydraulic Machinery and Systems, Lausanne, Switzerland