PIV measurements on a Kaplan turbine and comparison with scale-adaptive numerical analysis

Johannes Beck, Simon Jossberger, Roland Jester-Zürker and Stefan Riedelbauch

1 Voith Hydro Holding GmbH & Co KG, Alexanderstraße 11, 89522 Heidenheim, Germany
2 Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70569 Stuttgart, Germany
E-mail: Johannes.Beck@voith.com

Abstract.
Kaplan turbines build a mainstay of the hydro power generation as they offer best efficiency even for high flow conditions. A double regulating approach facilitates a high degree of flexibility but yields to be a sophisticated turbine type. Typical phenomena like gap flow and flow separation are challenges in simulation and complicate reliable predictions. To get a better understanding of flow physics elaborate numerical simulations by means of Computational fluid dynamics (CFD) are conducted as well as experimental flow investigation using the non-intrusive optical measurement technique Particle Image Velocimetry (PIV). Measurements are taken at the spiral case inlet, the draft tube cone and the diffuser giving instantaneous or phase-locked time averaged vector fields of flow velocity in a plane. Depending on the optical access it is possible to gather the in-plane velocity components or all three velocity components using Stereoscopic PIV. The obtained measurement results are then taken as reference on the one hand side for a definition of realistic inlet boundary condition and on the other hand side for assessment of the chosen numerical approach for predicting flow phenomena in Kaplan turbines. In this particular case the focus is on the interaction of runner and draft tube. Results of unsteady simulations with standard two equation and advanced hybrid RANS-LES turbulence models are compared to measurement data.

1. Introduction
CFD simulations act as the main important tool for the development of hydraulic machines, which is why there are strong ambitions to improve flow predictions in both industrial and academic environment. In order to achieve this goal detailed CFD simulations are conducted along with experimental flow measurements for reasons of validation. The focus of this work is on the preparation, conduction and evaluation of the measurements on a model machine test rig. Limited optical access of the machines often leads to restrictions in the choice of measurement setup (Planar PIV[1] or Stereo PIV[1]), the measurable area and the evaluability of data. The boundary conditions of the measurement location and it’s handling of challenges as well as the results of the intake and draft tube cone are shown in the following. In the last sections a short overview of the performed simulations and a first comparison between measurement and simulations are given.
2. Measurement

2.1. Overview measurement locations

Experiments are conducted at several locations in the model machine which are shown in Figure 1. The main measurement location is positioned in the draft tube cone to observe the runner outflow. The measurement data build the basis of comparison to CFD. Another area of interest is the flow behavior in the diffuser where temporal and spatial flow variation are assumed due to gap flow and flow separation. Additionally, a third measurement at the intake was carried out, to support a realistic modeling of the inflow conditions for the given semi spiral case.

Figure 1: Left: measurement positions overview. Measurements are conducted at the intake of the semi spiral case, at the draft tube cone and at the diffuser. Right: Measured operating points in a characteristic diagram. CFD simulations are based on OP1.

2.2. Operating points

Figure 1 shows numbered operating points (OP) in a characteristic diagram. The main measurement point OP1, for that the simulations were performed, is located close to rated head. Keeping the blade tilt $\varphi$ constant, two more points (OP4 and OP5) were measured at a higher and lower head at on-cam conditions. Besides that, OP2 and OP3 were measured at off-cam condition with a lower and higher gate-opening $\Delta \gamma$ related to OP1. Furthermore the optimum efficiency point OP6 was measured along with another point OP7, located at rated head, while keeping $\varphi$ of the optimum.

2.3. Measurement setup

As the velocity distribution in a measured section is of big interest, PIV was supposed to be the ideal measurement technique. Wherever possible Stereo PIV has been applied which offers the out-of-plane velocity component additionally to the in-plane velocity components of planar PIV. For reasons of illumination, a ND-YAG double-pulse laser with a wavelength of 532nm was used, whereby hollow glass spheres of about 10$\mu$m, which acted as seeding material of the water, reflected their light to be recorded by one or two 5MP SCMOS cameras. The whole PIV system allows recordings of 15Hz as a maximum. The model machine was equipped with a revolution counter, so velocity fields could be recorded in a phase-locked manner. Several parts of the model test machine were adapted, plexiglass windows inserted, to allow optical access. Planning and design of the windows, considering position and dimensions, was done by use of raytracing techniques, which allowed to predict viewing angles of cameras as well as illumination of measurement sections by casting single rays of the laser light sheet.

2.4. Recording and evaluation

The PIV measurement technique allows to determine a dense velocity field of a plane by capturing the individual velocity vectors simultaneously at the same time. As PIV is based
on particle images, a calibration is necessary by use of a defined calibration target, which later on allows to transform image coordinates and displacements in real world coordinates and velocities. Individual calibration solutions will be shown in section 2.5 and 2.6. The method of self-calibration[2] is used to improve measurement errors due to light sheet misalignments. Recordings of a time-series were taken to determine temporal averaged velocity fields in conjunction with it’s standard deviation, giving information about the fluctuations. Furthermore, higher statistics could be processed like reynolds stresses or turbulent kinetic energy[3]. These time-series were taken at a constant phase, observing the flow at a certain runner position, showing for example the wake of the runner outflow. On top of that, phase-locked measurements were repeated at discrete phase offsets of 8°, covering the range of a blade channel.

2.5. Spiral case inlet

Figure 2 depicts the intake of the semi spiral case, which is located in the head water tank and is divided by two piers. To investigate the inlet flow under different operating conditions, three measurement planes were chosen right before each of the three intake channels. Therefore three windows were installed at the top of the head water tank to give an optical access to the laser light sheet. The observation of the illuminated planes was done by cameras from the side, having a perpendicular view on the planes. The optical access was given by use of four glass windows from the side. Lenses of different focal length were used to cover the same area at the three planes, located at different distances. Due to the huge dimension it was decided to do planar PIV measurements to circumvent a complicated Stereo PIV calibration procedure. Instead it was sufficient to use small planar point patterns to determine the magnification factor, rather than performing a full calibration. An aluminium frame was attached directly in front of the inlet. Several small pieces of point patterns were mounted on the frame, shown by the two images in Figure 2. Two cameras were possible to measure at the same time, why it was not possible to measure instantaneous velocity fields at all four areas of a plane. Further information on the measurement are given in Figure 2.

Figure 2: Inlet Measurement: CAD pictures show PIV setup and measurement locations of one illuminated plane, the first photograph show the aluminium frame in front of the inlet, the second one shows one of the point patterns, as well as short information about the measurement at the right side.

Figure 3 shows an extract of measurement results of operating point 1, which are used to set up meaningful boundary conditions for the simulation (see chapter 3). The vector fields shown, are stitched together by 12 individual measurements (three planes, four windows). Each measurement gives a time-averaged vector field of the in-plane velocity, based on 2000 instantaneous vector fields, recorded at 6Hz.
Figure 3: Inlet Measurement: time-averaged vector field of inflow at OP1 (CFD), whereby the in-plane velocity, normalized by the mean inflow velocity, is shown by color code.

2.6. Draft tube cone
Measurements at the draft tube cone are always challenging because of a limited optical access. At the given machine, a semi spiral case allows just a view from the front, having a look through a plexiglass cone (Figure 4). That was used to insert a laser light sheet, as well as a camera to have a look at the illuminated section. A second optical access was realized by an additional window, located in the draft tube elbow for a second camera, having a Stereo PIV setup. A two plane calibration target, shown in Figure 4, had been manufactured, so there was no need to traverse a planar calibration target inside the machine. The evaluable area was about 90% of the cross section, limited by illumination and the overlapping camera views. Steep viewing angles and high image distortions led to problems in evaluation, so a differing procedure of dewarping and calibration was applied, shown in Chapter 2.7. Some results of the measurement are shown in section 4.

Figure 4: Cone measurement: CAD pictures show PIV setup and measurement locations, the photograph shows the two plane calibration target, as well as short information about the measurement at the right side.

2.7. Calibration and dewarping
As already mentioned in chapter 2.6, strong distortions prevented the usual calibration model to work. A very common approach is the use of an empirical model, in which a 3rd order polynom describes the transformation from image coordinates to world coordinates[1]. For most of the area, the polynom could describe a pretty good transformation, but in the most interesting regions, close to the wall, where strongest distortion occur, the deviations were way to strong leading to wrong evaluations. A way to come around that problem, was a "pre-dewarping" approach, which was applied before the actual evaluation as a preprocessing step.
The basic idea is, to do reduce the strong distortions in the calibration pictures as well as in the particle recordings - called ”pre-dewarping” - and handle all these preprocessed recordings as they would have been measured that way already. On base of the pre-dewarped images, the standard calibration model of the PIV software was sufficient.

The dewarping itself was performed by the PIV software, but needed to be fed by a ”deformation”-vectorfield, describing the displacements dx and dy for each pixel, which are needed to come from the distorted to a dewarped image. The main task was to create this vector field. Figure 5 shows the individual steps. First the raw calibration image is needed and the positions of the markers on it. After destination points, beeing the real world coordinates, are defined, some more points get extrapolated to ensure good dewarping even for the gaps between calibration target and wall, where no points are available. Then for each pair of points, raw and destination points, a vector with dx and dy is calculated. Finally these vectors are needed at each discrete pixel location, wherefore inter-/extrapolation is applied. For all inter- and extrapolations the geostatistical ”Kriging” method is used.

Figure 5: Image Dewarping: first two pictures show in- and outcome of the dewarping. The illustration on the right shows the individual steps of the dewarping algorithm.

3. Numerical setup
Two simulations with different turbulence models were conducted in ANSYS CFX 19.1 on the same block structured mesh consisting of around 20 million hexahedral elements. Turbulence is modeled either with the standard two equation Shear Stress Transport (SST) Model or by a hybrid RANS-LES Stress-Blended Eddy Simulation (SBES) with the SST model in the RANS region and the WALE model in the LES region [4].

The computational domain starts at the three intake channels leading to the semi spiral casing and ends downstream of the tail water tank of the test rig. Due to the rectangular shape and short length of the intake channels, the assumption of a developed pipe flow is not correct as inlet boundary condition. Because of that, the experimental data shown in section 2.5 were used to set physical valid boundary conditions. A preceding unsteady simulation of the head water tank including the semi spiral casing was performed. The results were validated with the experimental data at the spiral case inlet (cf. figure 3). Afterwards time averaged three dimensional distributions of the preceding simulation for the velocity components and turbulent properties were used as inlet boundary condition when simulating the whole turbine.

Between stationary and rotating subdomains a full 360° transient rotor stator interface with all five runner blades modelled is used. Time step size for the unsteady SST simulation is equal to 2° runner rotation. The simulation was initialized with a steady state solution. After a build up time of 20 revolutions, the phase and time averaged data was obtained for additional 30 revolutions. The final time step size of the SBES simulation corresponds to 0.25° runner rotation. This is necessary to reach an average CFL number smaller than 1 in the LES region to avoid excessive damping of the turbulence [5]. The simulation was initialized with the unsteady
SST simulation and a time step size of 2°. After 5 revolutions the time step size was ramped down over the course of 100 iterations to the final size of 0.25°. Following 5 additional revolutions, the phase and time averaging was carried out for about 15 revolutions.

4. Comparison of simulation and experiment

In the following, a short comparison between experimental data and results of the simulations on the measurement plane in the draft tube cone for a fixed runner position will be given. All velocities are normalized with the applicable maximum value of the measurement.

Figure 6 shows the deviation of the simulations to the measurement for the circumferential average of the axial and circumferential velocity as lines. Variance is depicted as vertical bars. To get mean values, the velocities and associated variances were averaged in radial bands. Solely locations where valid measurement data is available were used for averaging the numerical results. For the most part the axial velocity of the two simulations is very similar and in good agreement with the measurement. The biggest deviations occur for small and high radii.

While the variance of the simulations is very small for the most part, the SBES simulation exhibits a very high variance for radii less than 0.15. Regarding the circumferential velocity both simulations behave similarly with the exception of radii smaller than 0.3. The dent in the velocity predicted by the SBES simulation around radius 0.95 is due to an overestimation of the downstream influence of the guide vanes (cf. figure 7). The behaviour of the measurement is better captured by the SBES simulation for radii in the range [0.15, 0.3]. This changes at radii smaller than 0.15 significantly. In this region the SBES simulation exhibits a high deviation with a very high variance. For the SBES simulation, both axial and circumferential velocity and high variance for radii smaller than 0.15 indicate a highly unsteady flow downstream the hub. This result is not present in the SST simulation and measurement.

Figure 7 shows the normalized circumferential velocity. Even though the averaged velocities of the simulations shown in figure 6 are quite similar to a large extent, the results differ when looking at the whole measurement plane. While the main features of the runner outflow are captured by both simulations, the results of the SST simulation are much smoother than the ones obtained by the SBES simulation. The SBES simulation leads to a more detailed flow field with higher spatial variation. This is in better agreement with the measurement. Furthermore, the downstream influence of the guide vanes on the velocity in the outer region of the cone is only visible in the SBES simulation and measurement, albeit overestimated by the simulation.

To investigate if the differences between the simulations are solely an effect of the turbulence model or also influenced by the different time step sizes, further simulations with the same time step size are necessary. Furthermore, a more detailed evaluation and comparison of the results...
between the simulations and on the measurement planes in the draft tube cone and the two diffusers is to be performed.

Figure 7: Normalized circumferential velocity for one phase angle in the draft tube cone. Left: SST simulation, middle: SBES simulation, right: measurement.

5. Summary and outlook
An overview of the locations and setup of the measurement and operating points was given. The detailed process of measuring the spiral case inlet at three measurement planes with planar PIV and the draft tube cone at one location with stereo PIV was shown. Due to strong distortions in the draft tube cone a "pre-dewarping" was necessary. Furthermore, a first comparison between a standard SST and a scale-adaptive SBES simulation and the measurements have been shown. The hybrid RANS-LES simulations shows promising results compared to the standard simulation. However, further simulations and evaluations are necessary to isolate the effect of the turbulence model.

As indicated in this paper and shown by other authors, i.e. Jošt et al. [6], Krappel et al. [7] and Junginger and Riedelbauch [8], advanced approaches to treat turbulence can resolve a higher degree of flow features and greatly improve the simulation results for hydraulic turbomachinery. Because of that, additional simulations with the SBES approach on increasingly fine meshes are planned to investigate the flow in Kaplan turbines in greater detail and to improve the simulation approach for this type of turbomachinery. The performed measurements build a superb basis to assess and validate these simulations.

References
[1] Raffel, M., Willert, C.E., Wereley, S., Kompenhans, J. 2007 Particle Image Velocimetry - A Practical Guide Springer International Publishing
[2] Wienecke, B. 2015 Stereo-PIV using self-calibration on particle images Experiments in Fluids. 39. 267-280
[3] LaVision GmbH 2018 LaVision DaVis Product manual Release 8.4
[4] ANSYS Inc 2018 ANSYS CFX-Solver Theory Guide Release 19.1
[5] ANSYS Inc 2018 ANSYS CFX-Solver Modeling Guide Release 19.1
[6] Jošt D, Skerlavaj A, Lipej A 2014 Improvement of Efficiency Prediction for a Kaplan Turbine with advanced Turbulence Models Journal of Mechanical Engineering Vol. 60
[7] Krappel T, Riedelbauch S, Jester-Zürker R, Jung A, Flurl B, Unger F, Galpin P 2016 Turbulence resolving flow simulations of a francis turbine in part load using highly parallel CFD simulations 28th IAHR Symposium on Hydraulic Machines and Systems
[8] Junginger B, Riedelbauch S 2017 Numerical Analysis of a Propeller Turbine Operated in Part Load Conditions High Performance Computing in Science and Engineering '17