Experimental and numerical analysis of cavitation and pressure fluctuations in large high head propeller turbine

Matic Ocepek¹, Zlatko Peršin¹, Igor Kern¹, Vesko Djelić¹, Simon Muhić² and Andrej Lipej²

¹ VT Turbo d.o.o., Ljubljana, Slovenia
² University of Novo mesto, Faculty of mechanical engineering, Novo mesto, Slovenia

Email: matic.ocepek@vt-turbo.si

Abstract. In the last decades, Computational Fluid Dynamics (CFD) has played an important role in the numerical prediction of cavitation characteristics of different types of hydraulic turbines. Relative results of the characteristics curves for different hydraulic shapes can be predicted very accurately. Usually the prediction of the absolute values of various characteristics is still a challenge. In the paper, we would like to present the comparison between the results of the model testing and numerical analysis of the critical cavitation coefficient for propeller turbine and the results of pressure fluctuation for various operating regimes.

For the numerical analysis, the cavitation model is based on the barotropic state law which describes the density change when the pressure falls below the liquid saturation pressure. Barotropic means that the density depends only on the pressure. The model tests were performed according the IEC 60193 international standards.

The results of the research contain the diagrams with sigma break curves obtained with model test and numerical analysis for different operating regimes. Furthermore, a visual observation of the cavitation on the runner blades is also presented and compared with numerically obtained cavitation regions inside the computational domain. According to the pressure fluctuations full range of operating points has been analysed and are presented.

1. Introduction
Nowadays hydropower is still the most important renewable energy source. Numerous new power plants are under construction and there are many existing hydropower plants which should be refurbished and modernized for the next decades of reliable energy production with smooth operation. Some existing hydropower plants with bigger number of units and high head double regulated Kaplan turbines are converting to Francis or even propeller type of turbines (Kaplan type with fixed runner blades). Propeller turbines are more environmental and fish friendly, running without oil in runner head and with minimal gap clearances of the blades [1].

Advanced tools as CFD simulation, integrated with model testing should be used in order to achieve up to date performances with turbine peak efficiency over 95%, smooth running with acceptable pressure fluctuations and cavitation free operation.
Cavitation is an important physical phenomenon that affects the operation of various machines and devices. It can be treated experimentally [2] or by using different numerical methods. The use of CFD in the research of hydraulic machines was mostly for the flows without considering cavitation, but with different physical models. Numerical analysis of cavitation can be analyzed quite fast and accurately [3]. In numerical calculations, cavitation can be considered as a two-phase flow [4] [5] or we can use a simpler approach using a so-called barotropic model [6]. The model is simple and does not require powerful computing capacity.

Approach for analysis of cavitation depends on how much details we would like to analyze with the numerical method [7]. In reality, the bubble arises and travels downstream. In the area of a higher-pressure the bubbles collapse. The consequences of all these phenomena are various side effects such as noise, material erosion, pressure oscillations etc. If not all of the phenomena listed above are relevant, we can numerically deal with simple cavitation physical model. The barotropic model was used to analyze the cavitation characteristics of a high head propeller turbine. In the first part, we wanted to determine numerically the sigma break curves for different operating regimes and compare results with model tests. The second part of the paper deals with the comparison between experimental and numerical results of pressure fluctuations in the cone of draft tube. For numerical analysis, we used the NUMECA FINE / Open [8] software package.

2. Model test

Model testing was performed in the hydraulic laboratory of ČKD in Blansko, Czech Republic (Figure 1) on the new upgraded vertical test rig. Testing was carried out with high accuracy and repeatability, also in compliance with all the requirements, stated in the IEC 60193 standard [9]. Based on the results of this measurements, the properties of the prototype may therefore be determined quite accurately with appropriate scale-up procedures for all the parameters with similar conditions. Measured efficiency is well above the guarantees. Prototype diameter is 7.2 m and nominal power is about 106 MW.

![Figure 1: Turbine model installed on the test rig](image)

Cavitation on the model test is measured by gradually lowering draft tube pressure, while keeping head and wicket gate opening. Turbine characteristics (head, torque, discharge, revolution speed, draft tube pressure...) are measured after achieving stable conditions within one setting. Complete sigma break curves are presented (Figure 6, Figure 7 and Figure 8).

Measuring of the pressure fluctuations within the turbine is done by using piezoresistive absolute pressure transducers. Those sensors were installed at five locations on the model turbine: one at spiral case inlet, two at draft tube cone and two at draft tube elbow as shown on the scheme in Figure 2. Signals were acquired with 2 kHz sampling frequency and 16-bit resolution. Continuous time waveforms stored on the computer hard disk were least 30 seconds long. Software developed in LabVIEW was used to
record and analyze pressure signals. Appropriate filters and data processing methods were used to extract valuable information in the time and frequency domain. Sigma plant cavitation conditions were attained during the testing. Additionally, frequency spectrums with amplitudes in % of head and normalized on the rotational frequency n, were extracted from the sampled signals from all the sensors. Selected location (p_{p2}) transducer spectra is presented in Figure 3.

![Figure 2: Pressure fluctuations measurement locations](image)

![Figure 3: Pressure fluctuations experiment spectra for different dimensionless wicket gate openings a_0 at location p_{p2}](image)

3. CFD analysis

3.1. Computational model

The computational domain (Figure 4) consists of a distributor, a runner and a draft tube. The computational grid for the distributor and runner is structured (Figure 5), and the grid in the draft tube is unstructured. The total number of elements was around 10 million.

![Figure 4: Computational domain](image)

The high number of elements is not the only condition for the quality of calculations. Also other parameters are important. In Figure 5 (left) high orthogonality inside the runner mesh is presented. For
all CFD calculations low-Re k-ω SST turbulence model was used, where recommended \( y^+ \) should be lower than 10, which is achieved in our case (Fig. 5 – right).

3.2. Cavitation

CFD calculation on the computational domain is done by setting the operating conditions similar as done on the model test. For the cavitation analysis we used the barotropic law, which is used to describe the density variation in function of pressure from the liquid to the gas density. This model can only be used with incompressible fluids and when the derivative of the saturation pressure with the temperature is small. In our case, both conditions are met. Outlet pressure is set to some value lower than atmospheric pressure and calculation is done until convergence criteria are met. Gradually reducing the outlet pressure to near vacuum and repeating the calculations gives enough calculation points to draw a sigma break curve and compare it to model test curve. Calculation results for different dimensionless \( a_0 \) wicket gate openings are in Figure 6, Figure 7 and Figure 8. Efficiency is normalized to 1.
While calculating the cavitation we had a constant flow rates in all operating regimes, but near the critical cavitation number the flow rate had to be corrected (increased) so that we kept a constant head.

3.3. Pressure fluctuations

Transient CFD calculation is needed to predict the pressure fluctuations. Computational meshes with some \( a_0 \) wicket gate opening were generated for the regions where different level of pressure fluctuations were expected. It is important to predict regions of maximum value of pressure fluctuations and shape of the curve in dependency of the discharge.

Several rotations of the runner are needed, to ensure fully developed flow field. After full 10 rotations with 5° step (timestep 8.33e\(-4\) s) data for one rotation was collected. One pressure fluctuations location was selected (\( p_{p2} \)), where maximum values were expected. Diagram in Figure 9 shows the values of absolute total pressure relative to the head in dependency of time for calculated wicket gate openings for the selected location. Relative peak-to-peak comparison to the experiment is shown in Figure 10.

![Figure 9: Pressure fluctuations of the absolute total pressure, CFD](image)

![Figure 10: Peak-to-peak amplitudes of the pressure fluctuations, CFD results comparison to the experiment](image)

4. Discussion

Model test and CFD comparisons of the sigma break curves for different wicket gate openings \( a_0 \) are presented in Figure 4, Figure 5 and Figure 6. Experiment mostly confirms CFD results of the important cavitation phenomena like efficiency level and location of Thoma number dependent break location. Visually observed bubble generation location and cavitation cloud size is quite comparable to the numerically achieved ones also.
Pressure fluctuations peak-to-peak amplitudes from CFD and model test results in the dependency of dimensionless discharge are normalized to their maximum values at \( a_0 = 1.0 \). They are presented as a relative values in Figure 10. In Figure 3, FFT analysis of wide band frequency signal from the model test data is shown. At best efficiency point (BEP) wicket gate opening \( a_0 = 1.6 \) pressure fluctuations amplitudes are negligible. When closing the wicket gate opening to \( a_0 = 1.4 \) and \( 1.2 \), partial load vortex becomes more intensive, therefore low frequency fluctuations of the vortex and its harmonics becomes dominant. Further reducing the wicket gate opening to \( a_0 = 1.0 \) and 0.8 yields the dominant frequency of 100 Hz which is exactly the blade passing frequency (BPF) of the 6 blade runner rotating at 1000 rpm. In comparison the results from CFD analysis of the pressure fluctuations detects only BPF frequencies and its first harmonics (Figure 9) because of the short dataset.

5. Conclusions
Prediction of the cavitation phenomena during the design and optimisation phase can be quite reliably done using only CFD calculations instead of model testing, which is considerably more expensive and time consuming.

Pressure fluctuations analysis shows, that when BPF is a dominant frequency on the model, the comparison of the relative peak-to-peak amplitudes to the CFD results indicates a good similarity. This is valid for the wicket gate openings \( a_0 = 0.8 \) and 1.0. In other cases this relation is not so clear anymore.

Frequencies lower than BPF are only indicated from the CFD results, because in this case the time waveform is not long enough for proper evaluation. To further improve CFD results and capture also lower frequency phenomena, number of calculated runner revolutions should be increased to lengthen the dataset.

References
[1] M. Odeh, A Summary of Environmentally Friendly Turbine Design Concepts, U.S. Department of Energy Idaho Operations Office, 1999
[2] M. Grekula and G. Bark, “Experimental Study of Cavitation in a Kaplan Model Turbine,” CAV2001, no. 1997, pp. 1–8, 2000
[3] D. Jošt, A. Lipec, P. Mežnar 2008 Numerical prediction of efficiency, cavitation and unsteady phenomena in water Turbines, 9th Biennial ASME Conf. on Eng. Sys. Design and Analysis, ESDA08 (Haifa, Israel)
[4] P. J. Zwart, A. G. Gerber, and T. Belamri, “A Two-Phase Flow Model for Predicting Cavitation Dynamics,” ICMF 2004 Int. Conf. Multiph. Flow, no. 152, 2004
[5] D. Jošt, M. Morgut, A. Škerlavaj, and E. Nobile, “Cavitation Prediction in a Kaplan Turbine Using Standard and Optimized Model Parameters,” 6th IAHR Int. Meet. Workgr. Cavitation Dyn. Probl. Hydraul. Mach. Syst. Sept. 9-11, 2015, Ljubljana, Slov. CAVITATION, 2015
[6] O. Coutier-Delgosha, R. Fortes-Patella, B. Stoffel, J. L. Reboud, and M. Hofmann, “Experimental and Numerical Studies in a Centrifugal Pump With Two-Dimensional Curved Blades in Cavitating Condition,” Trans. ASME, Vol. 125, 2003
[7] D. Valentín, A. Presas, M. Egusquiza, C. Valero, and E. Egusquiza, “Transmission of High Frequency Vibrations in Rotating Systems. Application to Cavitation Detection in Hydraulic Turbines,” pp. 1–18, 2018
[8] “Numeca International FINE/Open Theory Guide,” 2017
[9] IEC 60193 (1999-11), Hydraulic turbines, storage pumps and pump-turbines - Model acceptance tests, ISBN: 2-8318-4993-4 – ICS code: 27.140 – TC 4 – 569 pp.