Numerical and experimental research on unsteady cavitating flow around NACA 2412 hydrofoil

M Sedlář, M Komárek, P Rudolf, J Kozák and R Huzlík

1 Centre of Hydraulic Research, J. Sigmunda 190, 783 49 Lutín, Czech Republic
2 Brno University of Technology, Faculty of Mechanical Engineering, Technická 2896/2, Brno, Czech Republic
3 Brno University of Technology, Faculty of Electrical Engineering and Communications, Technická 3082/12, Brno, Czech Republic

E-mail: m.sedlar@sigma.cz

Abstract. This work deals with the numerical and experimental investigation of unsteady cavitating flow around a prismatic NACA 2412 hydrofoil. The main attention is focussed on the dependence of cavitation dynamics on the cavitation number at high incidence angles. The experimental research is carried out in the cavitation water tunnel the rectangular test section of which has inner dimensions 150×150×500 mm. Currently tested hydrofoils have a chord length of 120 mm and are equipped with pressure transducers at the leading edge and on the suction side. The PVDF hydrophone enables to measure high-frequency pressure pulses behind the hydrofoil trailing edge. The visualizations are based on two simultaneous high-speed cameras, recording the hydrofoil from the top and from one side. A comprehensive CFD analysis has been done with the ANSYS CFX package for a wide range of flow regimes. Different turbulence models including SAS-SST and Reynolds-stress models have been tested to capture highly unsteady phenomena on the hydrofoil. The numerical simulations show, that the dominant frequency of the cavity oscillation depends on the cavitation number and that there is a certain range of this number in which the “resonance” effect can be reached. In such regime the amplitudes of the pressure pulses on the suction side of the hydrofoil dramatically increase. The calculated results have been verified by both the visualizations and the pressure measurements carried out at the hydrofoil incidence angle of 8 degrees.

1. Introduction

Hydrodynamic pumps are very often operated in the cavitation regime. Under such conditions the pump can suffer from cavitation instabilities resulting in unwanted noise and dangerous vibrations. The cavitation instabilities have been studied intensively in the case of inducers which represent a special type of axial-flow pumps with very small incidence angles. Most of the work is based on experimental research. One of the diagnostics which is used to determine the type of cavitation instability is the frequency analysis. We have to examine the dominant frequencies and compare them with the rotational frequency \( f_\Omega \) as well as with the fundamental frequency of the inducer blade passage excitation. Typical frequencies of some cavitation instabilities in inducers can be found e.g. in [1]. Recently some studies related to the cavitation instabilities in pumps have appeared [2-5] which

---

4 Corresponding author.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
are based mainly on numerical simulations. In [5] it has been shown that cavitation instability in the backflow vortices in front of the pump impeller appears in a limited interval of the cavitation number which is caused by the interaction of the separated flow and cavitation regions. When the cavitation number is sufficiently high, cavities formed on the impeller blades are small and relatively stable. On the other hand when the cavitation number is very low, the cavitation regions remarkably block the blade-to-blade passages of the impeller and the backflow practically vanishes.

To study experimentally interaction of cavitation structures with separated flow inside the complex geometry of a real pump is highly expensive and time-consuming and in some cases practically impossible. That is why the experimental research with a simplified geometry (prismatic NACA 2412 hydrofoil) in the cavitation tunnel has been initiated in the Centre of Hydraulic Research. This experimental research is accompanied with the numerical modelling.

2. Experimental set-up

The cavitation tunnel in the Centre of Hydraulic Research was built in 2007 especially for the investigation of erosion process triggered by cavitation [6]. The facility is a horizontal plane water tunnel for isolated hydrofoils. The closed loop is equipped with both compressor and vacuum pumps capable of creating different pressure levels while maintaining constant volume flow rate and includes the main tank with the capacity of 35 m$^3$. The rectangular test section has inner dimensions 150×150×500 mm and is made of organic glass to facilitate visualization from all sides (figure 1). The maximum tunnel inlet velocity 25 m/s corresponds to the maximum flow rate of the variable-speed driven axial-flow pump.
The currently tested hydrofoils are prismatic with a chord length of 120 mm, which gives the span/chord ratio higher than 1 (namely S/C=1.25). The span/chord ratio influences the shape of the shedding cavities as it will be shown in following sections. The profile incidence angle can vary between 0° and ± 180°. The NACA 2412 hydrofoil has been chosen as the test case as it is close to the hydrofoils used in axial-flow and mixed-flow pumps.

The main attention is focussed on the dependence of cavitation dynamics on the cavitation number at high incidence angles. Several incidence angles have been considered nevertheless the main part of the work is devoted to the regimes with the incidence angle of 8°, which is the same as used in reference [7], though its authors used different hydrofoil (Clark Y) and different span/chord ratio (S/C=0.81). Two types of hydrofoils have been designed. The first one is equipped with pressure transducers at the leading edge and on the suction side. The second type is covered with a grid of PVDF films which enable to monitor forces resulting from violent collapses of the cavitation structures. However the results presented in this study have been obtained with the hydrofoil with only two pressure transducers at midspan, at the leading edge (pressure P0) and on the suction side (at 40% of the chord length, pressure P40), which minimizes the stimulation of cavitation by the tap holes (figure 2). The PVDF hydrophone enables to measure high-frequency pressure pulses behind the hydrofoil trailing edge (105% of the chord length). The pressures have been measured with the sampling frequency of 5 kHz, 10 seconds each record. The accuracy of the transducers is 0.5 kPa. The visualizations are based on two simultaneous high-speed cameras, recording the hydrofoil from the top and from one side. Each video-record represents two seconds with the frame rate of 1000 fps and a high resolution 1024×1024 pixels. The pressure measurements are synchronized with both cameras.

![Figure 2. Scheme of test section and hydrofoil with two pressure transducers.](image)

### 3. CFD tools and simulation details

In this study the ANSYS CFX package has been used to solve the URANS equations. The Rayleigh-Plesset model included in the ANSYS CFX package has been applied to describe the interphase mass transfer in the framework of the homogenous multiphase model. Though this cavitation model is based on a highly simplified Rayleigh-Plesset equation for the radius of a spherical cavitation bubble, it is quite robust and effective for problems, where the void fraction is highly important in forming flow structures but where the details of the bubble dynamics (especially the collapses and rebounds) are not of primary interest [5]. The calculations have been performed both with a full tunnel geometry (including the inlet contraction chamber, outlet diffuser and a part of the piping) and using a simplified geometry consisting of only the test section and a rectangular extension. The differences resulting from this simplification seem to be negligible, as the velocity uniformity at the front part of the test section is (based on the CFD analysis) within ± 1%, excluding the boundary layer. The structured computational grid of the test section itself represents approximately 2 million nodes in the case of the standard geometry with the span/chord ratio S/C=1.25. The computational time step has been set in the range 1-5×10⁻⁴ s depending on the turbulence model used. The boundary conditions are based on
prescribed total pressure at the inlet and the mass flow rate at the outlet. No symmetry plane at the hydrofoil midspan has been used.

To capture the highly unsteady phenomena close to the hydrofoil the widely used SST turbulence model has been tested as well as advanced turbulence models implemented in the ANSYS CFX code, namely the BSL based Reynolds-stress model, the SSG Reynolds-stress model and the SAS-SST model. No corrections of these models related to cavitation have been done.

4. Results and discussion

A comprehensive CFD study has been carried out for a wide range of flow regimes. As the main attention is focussed on the dependence of cavitation dynamics on the cavitation number, the results for a constant Reynolds number of $1.55 \times 10^6$ are discussed in this paragraph. It should be noted that the dimensionless numbers (like Reynolds and cavitation ones) in this study are based on the hydrofoil chord length and the test section inlet velocity.

The numerical simulations show, that the dominant frequency of the cavity oscillation depends strongly on the cavitation number and that there is a certain range of this number in which the “resonance” effect can be reached. In such regime the amplitudes of the pressure pulses on the suction side of the hydrofoil dramatically increase. Figure 3 shows two dominant frequencies obtained by CFD and experimentally, as a function of the cavitation number $\sigma$, and the amplitudes of the first dominant frequency. In the CFD analysis sheet cavities formed on the hydrofoil are small and relatively stable when $\sigma$ is sufficiently high. There is only one dominant frequency about 21 Hz ($St \approx 0.2$) and amplitudes are very low. With decreasing cavitation number we can identify two (or more) significant frequencies. The lowest one (about 15 Hz, which corresponds to $St \approx 0.14$) is the most significant. In this regime we can see dramatically increasing amplitudes. Further drop of $\sigma$ causes decrease of the amplitudes and in the end, for sufficiently low values of $\sigma$, a stable supercavitation regime starts in which it is impossible to detect any dominant frequencies. In this regime the backflow on the suction side of the hydrofoil practically vanishes. Experimentally obtained data show the same tendencies but it can be seen that the “resonance” region is quite narrow compared to the CFD results which makes it difficult to detect. The result of the FFT analysis of calculated static pressure $P_{40}$ in the regime of “resonance” can be seen in figure 4.

![Figure 3](image)

Figure 3. First two dominant frequencies (left) and pressure amplitudes of the first dominant frequency (right). Comparison of CFD and experiment.

Graphs in figure 3 are based on the normalised cavitation number (related to the cavitation number $\sigma_{Res}$ in which the maximum “resonance effect” appears) and on the normalised amplitudes of the static pressure $P_{40}$. The reason is that in the case of experiments the “resonance” cavitation number $\sigma_{Res}$ changes according to the immediate nuclei content in water. In presented calculations $\sigma_{Res}$ has a value of about 1.5; in measurements it is in the range approximately from 1.6 to 1.75. On the other hand,
values of dominant amplitudes depend on different methodology of evaluation of experimental and numerical results. Nevertheless an agreement of the calculated and measured pressure $P_{40}$ in the same regime ($\sigma = \sigma_{\text{Res}}$) can be seen in figure 5.

To verify the agreement of the pressure measurements and visualizations, the image analysis of movie sequences has been carried out in several points close to the position of the pressure transducer $P_{40}$ tap hole. Values from 1 to 256 were assigned to selected pixels according to the gray scale and the FFT analysis has been applied (figure 6). The transformation of the temporal image sequence to frequency domain provides the amplitude-frequency spectra, which identify dynamic behavior of the main coherent structures of the separated cavitating flow (figure 7). The image processing agrees very well with the results of CFD analysis as well as the pressure measurements for both the individual points and the point set average.

Another parameter which can affect the unsteady cavitating flow around the hydrofoil and which has been examined is the span/chord ratio. As it is very complicated to change dimensions of the test chamber in the cavitation tunnel, this investigation has been done only in a numerical way. The simplified geometry consisting of the test section itself with a rectangular extension has been applied. The span/chord ratios 0.417, 0.625, 1.25 and 1.875 have been considered. In addition a case of an infinite span has been modelled using the periodic boundary conditions on the side walls. Here only SST turbulence model and the Reynolds-stress models could be used, as the periodic (or symmetry) boundary conditions do not fit the SAS-SST turbulence model. The CFD analysis has shown that the span/chord ratio can influence the shape of the cavitation structures (figure 8) but the basic topology of the flow separation and the dominant frequencies and their amplitudes are influenced only a little. For example in the vicinity of the “resonance” cavitation number $\sigma_{\text{Res}}$, the first dominant frequency changes from the value of 13.85 Hz (S/C=0.417) to the value of 14.83 Hz (infinite S/C).

All results presented in this section have been obtained with the SAS-SST turbulence model except the case with the periodic boundary conditions where the BSL Reynolds-stress model has been used.

**Figure 4.** FFT analysis of static pressure $P_{40}$. CFD, “resonance” cavitation number $\sigma_{\text{Res}}$.

**Figure 5.** Dynamics of static pressure $P_{40}$. 3 cycles. Experiment and CFD, $\sigma = \sigma_{\text{Res}}$. 
5. Conclusions
Highly unsteady cavitating flow around the NACA 2412 hydrofoil has been examined both numerically and experimentally for different flow regimes, especially for a wide range of cavitation numbers. The comparison of obtained data shows that the CFD analysis is able to predict main flow and cavitation structures (see figure 7 and figure 8 left) and numerical results agree reasonably with experiments from the quantitative point of view.

Visualizations and especially CFD analysis have provided a deep look at the development of the flow and cavitation structures around the hydrofoil in time. Unfortunately there is not enough space to present these structures in detail in this paper.

Different turbulence models have been tested during the project. Among them, the SAS-SST model has demonstrated superior results. Nevertheless future work tends to highly detailed analysis with very fine computational grids and non-URANS turbulence models like LES or DES simulations. Very promising is also the PANS model, successfully used (in the framework of the ANSYS CFX code) in [8] for very detailed analysis of flow and cavitation structures around a Delft twisted hydrofoil.

[Images of charts and diagrams related to flow analysis and cavitation regions are shown.]

**Figure 8.** Separation of cloud cavities in back-flow region (top row) and surface streamlines (bottom row) on hydrofoil suction side. CFD, $\sigma = \sigma_{Res}$. Cavitation regions visualized as locations where void fraction exceeds 10%.

**Acknowledgements**
This work has been supported by the grant no. 101/13/23550S of the Czech Science Foundation, entitled “Experimental Research and Mathematical Modelling of Unsteady Phenomena Induced by Hydrodynamic Cavitation”.

References

[1] Tsujimoto Y 2007 Cavitation instabilities in turbopump inducers Fluid Dynamics of Cavitation and Cavitating Turbopumps ed dAgostino L and Salvetti M V (Vienna: Springer-Verlag) pp 169-90

[2] Yamanishi N, Fukao S, Qiao X, Kato C and Tsujimoto Y 2007 J. Fluid Eng. 129 587-94

[3] Yamamoto K and Tsujimoto Y 2009 Int. J. Fluid Mach. and Systems 2 40-54

[4] Kobayashi K and Chiba Y 2010 Int. J. Fluid Mach. and Systems 3 113-21

[5] Sedlář M, Šputa O and Komárek M 2012 Int. J. Fluid Mach. and Systems 5 18-29

[6] Sedlář M, Komárek M, Vyroubal M and Müller M 2012 Experimental and numerical analysis of cavitating flow around a hydrofoil EPJ Web of Conf. (Madrid, Spain 25-27 June 2012) 25 01084

[7] Watanabe S, Konishi Y, Nakamura I and Furukawa A 2011 Experimental analysis of cavitating behavior around a Clark Y hydrofoil Proc. WIMRC 3rd Int. Cav. Forum (Warwick, UK, 4-6 July 2011)

[8] Ji B, Luo X W, Peng Y X and Duan Y 2013 Int. J. Multiphase Flow 51 33-43