Experimental and numerical investigation of cavitation on Clark Y foil

D Homa1, 2, W Wróblewski1, M Majkut1 and M Strozik1

1Institute of Power Engineering and Turbomachinery, Silesian University of Technology, Konarskiego Street 18, Gliwice, Poland
2To whom any correspondence should be addressed

Abstract. The results of experimental investigation of cavitation in flow over Clark Y foil are presented. The cavitation test rig included a chamber with a blade profile fixed to the rotary disc that enabled to set different angles of attack. Working fluid was water at temperature 16°C. For different flow conditions (water velocity, pressure) the pictures of cavitating flow were taken and analyzed. The pressure at the inlet and outlet of the chamber were measured, as well as the value of volumetric flow rate. That enabled to determine the cavitation number for each case. The mechanism of cavitation structures appearance, growth and collapse was observed and described. The numerical simulation of cavitating flow was also performed by means of OpenFOAM software with interphaseChangeFoam solver. Kunz cavitation model was chosen for calculations, with k-ω SST turbulence model. The assumption of isothermal, two–phase flow was made. The vapour areas appearance, their shapes and changes in time were described and compared with experimental results. The main features of cavitating flow were caught, but the further adjustment of cavitation model is recommended.

1. Introduction
Cavitation is a complex process that includes both vapour bubbles formation in the flow and their sudden collapses. Cavitation can lead to many unwanted effects such as noise, vibrations, machine’s efficiency drop and erosion [1]. There are different types of flow with cavitation, but as far as turbomachinery performance is concerned, the main interest is focused on the flow over a hydrofoil. Many studies have been devoted to experimental investigation of cavitating flow [2-4] which provide substantial information about cavitation inception, development and dynamics. Especially in case of erosion risk assessment, it is necessary to identify locations of bubbles rebounds and collapses [4]. An important part of the experimental investigation is the registration of characteristic features of the cavitating flow, such as induction of reentrant jet or cavitating clouds detachment. Capturing high frequency cyclic changes of the cavitation structures demands high speed video camera and appropriate illumination system. Due to the rapid progress in computer science and computational fluid mechanics, it is recently more popular to simulate cavitating flow by means of CFD programs. Simulation performed in OpenFOAM is also included in this study.

The flow parameter which describes flow susceptibility to cavitation is called cavitation number σ [5]:

\[ \sigma = \frac{P_e - P_v}{0.5 \rho V_c^2} \] (1)

\[ \frac{\rho \Omega^2}{\gamma} \]
where: $p_\infty$ – free stream pressure, Pa; $p_s$ – saturation pressure, Pa; $\rho$ – liquid density, kg/m$^3$; $u_\infty$ – free stream velocity, m/s.

The lower cavitation number the more intense cavitation should appear in the flow.

2. Materials and methods

2.1. Experimental investigation

To investigate cavitating flow over a foil a cavitation tunnel was built at the Institute of Power Engineering and Turbomachinery at Silesian University of Technology. The test rig included rectangular chamber with the blade fixed to the disc that enabled to set different angles of attack. The investigation concerned Clark Y 11.7 % foil of chord (c) equal to 70 mm. The height of the chamber was 189 mm (2.7c) and the span (width) of the chamber was 70 mm (1c). The foil was placed at the half of the chamber’s height, 210 mm downstream from inlet of the chamber. The overall length of the chamber was 700 mm. At the top, bottom and one side wall of the test section transparent acrylic glass was used to enable observations. The scheme of the circuit is shown in figure 1. The 3D model of the test rig is depicted in figure 2. The measurements of static pressure at the inlet and at the outlet of chamber were performed, as well as the temperature at the outlet. Volumetric flow rate was determined by means of electromagnetic flowmeter. On the basis of flowmeter record the free stream velocity was determined. That enabled to calculate the cavitation number for each case. The measurement signals were collected by the computer unit. By means of pump’s motor variable frequency drive the rotational speed of pump was changed and the flow rate was increased. These caused the pressure in the chamber to drop and more intense cavitation to occur. During the measurements the movies of cavitating flow were taken by means of high speed camera Phantom Miro C110. The recording speed was set to 1200 frames per second, which was enough to capture the changes during one cycle of cavitation forming and collapse, but also sufficient spatial resolution of frames equal to 1280 x 720 pixels was kept. The series of measurements were performed for three different angles of attack, in the range of cavitation numbers from 1.5 (cavitation inception) to 0.75 (cloud, developed cavitation). The experiment was carried out on water of temperature equal to 16°C. The feeding tank had about 35 m$^3$ capacity, which enabled to keep water temperature at the constant level during the series of measurements. After experiment, the movies were split into series of frames. The period of changes for each case was estimated. The visualization included two different camera views, so that changes in spanwise and streamwise directions could be observed. It should be noted that only one camera was used in the experiment, but cyclic character of changes enable to match top and side view pictures. The view of recording equipment is shown in figure 3.

![Figure 1. The scheme of the circuit. 1 – water tank, 2 – pump, 3 – flowmeter, 4 – cross section reduction, 5 – chamber with blade 6 – computer unit. P – pressure measurement, T – temperature measurement](image1)

![Figure 2. The scheme of the test rig. 1 – inlet pipe, 2 – cross section change, 3 – measurement chamber, 4 – blade, 5 – outlet pipe.](image2)
2.2. Numerical investigation

For modelling the process of cavitation Kunz model was chosen. It was introduced by Kunz et al in [6]. It is a model of one-fluid models group. In the one-fluid model the flow is assumed to be a mixture of two phases and the conservation equations (mass and momentum) for the mixture are to be solved. In this simulation no slip between the phases was assumed. To calculate fraction of gaseous phase mass conservation equation of vapour is solved [7]:

\[
\frac{\partial \alpha}{\partial t} + \nabla (\alpha \rho \mathbf{u}) = \begin{cases} R_c & p \leq p_s \\ \alpha \rho \mathbf{u} \cdot \mathbf{n} & p > p_s \end{cases}
\]

(2)

where: \(\alpha\) – vapour volume fraction, \(\rho\) – vapour density, \(\mathbf{u}\) – velocity, \(R_c\), \(R_s\) – source terms, kg \(\cdot\) m\(^3\) \(\cdot\) s\(^{-1}\).

In Kunz model the source terms are derived empirically [6]:

\[
R_c = \frac{C_s \rho_l \alpha \min(0, p - p_s)}{0.5 \rho_l u_\infty^2 t_\infty}
\]

(3)

\[
R_s = \frac{C_s \rho_s \alpha^2(1 - \alpha)}{t_\infty}
\]

(4)

Where: \(\rho_l\) – liquid density, kg \(\cdot\) m\(^3\); \(u_\infty\) – free stream velocity, m \(\cdot\) s\(^{-1}\); \(t_\infty\) – mean flow timescale, s; \(C_s\), \(C_e\) – empirical coefficients.

In the domain the foil was placed in the same position as it was during the experiment. The 2D structural grid was generated and extruded in perpendicular direction to the overall width of 0.9 mm (3 layers of 0.3 mm thickness each). The geometry has been divided into 8 blocks and the O-grid was generated around the blade. The blade profile was split into 4 edges: near leading edge, upper side, lower side and trailing edge. On lower and upper side edges 90 grid nodes were set, on the leading and trailing edges – 45 nodes. On the edge normal to the foil 81 elements were set. The whole mesh consisted of 160k hexahedra elements. The overview of the mesh with the zoom of the O-grid region is shown in figure 4.
At first calculation with high cavitation number (single phase flow) was performed. The pressure distribution along the foil was monitored and compared with results obtained at Kyushu University, Japan [3] for the same foil. Pressure coefficient $C_p$ is defined as:

$$C_p = \frac{p - p_{\text{inlet}}}{0.5 \rho u^2}$$

(5)

The pressure coefficient distribution along the foil is depicted in figure 5. The generated mesh gave good agreement with the experimental results, so it was chosen for further computations.

The simulation was performed with inlet velocity and outlet static pressure boundary conditions. The side planes of the domain were set to symmetry conditions, upper and lower planes of the domain - slip wall condition. The pressure at the outlet was consequently lowered and velocity increased, according to the values obtained in the experiment. The cavitating flow was simulated by means of OpenFOAM software. The interPhaseChangeFoam solver that was used allows the phase change between two incompressible, isothermal, immiscible fluids. The assumption of vapour incompressibility has been reported in other investigations with satisfying results [8, 9]. Flow was assumed to be isothermal. The turbulence model was k-$\omega$ SST with turbulence intensity at the inlet equal to 5%. The transient, first order Euler scheme was chosen. For single phase calculations the time step of value $10^{-5}$ s was set. For cavitating flow simulations the constant time step was used, set to the value that ensured max Courant number under 0.4. The coefficients $C_e$ and $C_c$ in Kunz model were set to the value of 1000 [3]. During the calculations maximum value of $y+$ parameter on the foil wall was 10 (average value was about 2).

3. Results

3.1. The influence of angle of attack

Four different levels of cavitation number were recorded: 1.3; 1.0; 0.85 and 0.75 with three different
angles of attack (AoA): 4°, 6°, 8°. The movies were recorded and split into series of frames. On the basis of frame analysis the period of changes was estimated. In figure 6 the case of incipient cavitation is shown ($\sigma = 1.3$, $\alpha_a = 4^\circ$). The estimated shedding frequency for this case was 39 Hz. In figure 6A and 6B the moment of cavitation structure appearance is caught. Cavitation structure was first observed near the contact line of the foil and the rotary disc. It enlarges (6C, 6D, 6E, 6F) and eventually the structure detached from the foil, while a new structure started to grow from the leading edge (6G, 6H). In figure 7 the developed, cloud cavitation is shown with $\sigma = 1.29$ and $\alpha_a = 8^\circ$. It can be observed that angle of attack had significant impact on obtained cavitating flow pattern. The structures became larger and occupied the whole width of the blade. The frequency of changes was about 47 Hz. The estimation of frequency value was difficult due to the fact, that in this case there was no evident moments of structure collapse and growth of a new structure, it happened simultaneously. The example of this is shown in picture 7C and 7D, where the former structure collapses after the detachment, while a new structure reaches about 1/3 of chord length.

Figure 6. Incipient cavitation, $\sigma = 1.29$, $\alpha_a = 4^\circ$, $u_\infty = 9.8$ m/s, $p_m = 63$ kPa; left column – side view, right column – top view
The analysis of the top view images showed the different shape of the vapour structures in the streamwise direction. Up to the 20% of chord length was occupied by attached packet of bubbles, then the structures detached from the foil wall and cloud of tiny bubbles can be observed (Fig. 7D). The cloud moved to the rear region of the blade (Fig. 7E, 7F) and eventually collapsed (Fig. 7G, 7H).

3.2. Comparison of experiment and CFD simulation
During the simulations the vapour volume fraction in the computational domain was monitored. Then the frequency of changes was determined and one period selected to images analysis. In figure 8. the change of vapour volume fraction in time is presented for case $\sigma = 1.29$ and $\alpha_a = 8^\circ$, described in chapter 3.1. The frequency of changes was 17 Hz, less than was observed in the experiment. The period selected to visualization is depicted in figure 9 with locations of specific moments shown in figure 10. The comparison of flow pattern obtained in simulations and in the experiment is included in
Figure 8. Vapour volume fraction in domain, $\sigma = 1.37$, $\alpha_a = 8^\circ$

Figure 9. The chosen period of changes for $\sigma = 1.37$, $\alpha_a = 8^\circ$

Figure 10. Developed cavitation, $\sigma = 1.37$, $\alpha_a = 8^\circ$, $u_\infty = 9.7$ m/s, $p_{in} = 65.7$ kPa; left column – experiment, right column – numerical simulation
figure 10. Structures in the experiment were higher and longer than the ones obtained in calculations. In numerical simulation the structure grew (Fig. 10B), then split into two clouds (Fig. 10D) – the one closer to leading edge shrank (to about 40% of chord length), the other detached from the foil, moved to the trailing edge and collapsed (Fig. 10F). Meanwhile the structure near the leading edge enlarged to about 60% of chord length (Fig. 10H) and split again. During the experiment the structures appeared simultaneously, which can be seen when comparing images 10A and 10B. During the simulation there were moments when only one cloud existed, contrary to the experiment.

4. Summary and conclusions
In the paper the results of both experimental and numerical investigation of cavitating flow over Clark Y foil are presented. The visualization of cavitation appearing in the flow in both side and top views was done. That provided necessary information about shape and growth of 3D cavitation structures. During the top view frames analysis it was noticed that the structures exhibited non-uniformity in spanwise direction. The longest vapour structures occurred near the contact line of the blade and the disc that it was placed on. It was especially visible in case of low cavitation number and AoA cases. With increase of cavitation number and AoA the vapour clouds covered whole width of the blade. The phenomenon of recurring growth, detachment from foil wall and collapse of the structures near the blade rear region was observed and recorded. On the basis of image analysis also frequency of changes was estimated. For cavitation number equal to about 1.3 it was 39 Hz ($\alpha_a = 4^\circ$) and 47 Hz ($\alpha_a = 8^\circ$). The numerical investigation included simulation of the flow with parameters obtained during the experiment, with Kunz cavitation model. Although some simplifications were assumed, the obtained flow pattern was similar to the one from the experiment, the characteristic features of cavitating flow were noticed. The frequency of changes was lower in the numerical results, it was equal to 17 Hz. That leads to conclusion that selected cavitation model can be used to simulate this type of flow, but some modifications are recommended to improve accuracy of predicted shedding frequency and outreach of vapour structures.

5. References
[1] Brennen C E 1995 Cavitation and bubble dynamics (Oxford: Oxford University Press)
[2] Wan G, Senocak I, Shyy W, Ikohago T and Cao S 2001 Dynamics of attached turbulent cavitating flows Progress in Aerosp. Sciences vol 37 pp 551-81
[3] Matsunari H, Watanabe S, Konishi Y, Suefuji N and Furukawa A 2012 Experimental/numerical Study on Cavitating Flow around Clark Y 11.7% Hydrofoil Proceedings of the Eight International Symposium on Cavitation CAV2012
[4] Sedlar M, Komarek M, Wyroubal M and Muller M 2012 Experimental and numerical analysis of cavitating flow around a hydrofoil EPJ Web of Conferences vol 25 01084
[5] Franc JP and Michel JM 2005 Fluid Mechanics and its Application vol 76 (Fundamentals of cavitation), ed Kluwe Academic Publishers
[6] Kunz R 2000 A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction, Computers & Fluid Vol 29 p. 849-875, 2000
[7] Puffary B 2006 Numerical modelling of cavitation, design and analysis of high speed pumps Educational Notes RTO-EN-AVT-143 paper 3, pp 1–54
[8] Huang R, Luo X and Ji B 2017 Numerical simulation of the transient turbulent flows around Clark-Y hydrofoil using modified partially averaged N-S methods Journal of Mechanical Science and Technology vol 31(6)
[9] Capurso T et al. 2017 Numerical investigation of cavitation on a NACA0015 hydrofoil by means of OpenFOAM Energy Procedia 126 pp 794 – 801

Acknowledgments
The presented work was supported by the Polish National Science Centre founds within the project UMO-2016/21/B/ST8/01164.