The features of cavitation flow around a cylinder with a very low aspect ratio

S G Skripkin\textsuperscript{1,2,*}, M A Tsoy\textsuperscript{1,2}, A Y Kravtsova\textsuperscript{1,2}

\textsuperscript{1}Kutateladze Institute of Thermophysics, Siberian Branch of the Russian Academy of Sciences, Lavrentyev Avenue 1, Novosibirsk 630090, Russia
\textsuperscript{2}Department of Physics, Novosibirsk State University, Pirogova Street 1, Novosibirsk 630090, Russia

E-mail: skripkin.s.g@gmail.com

Abstract. The arising of the cavitation cavity behind the circular cylinder with aspect ratio 0.06 has been investigated using high-speed visualization. High-speed imaging at a sampling rate up to 200 kHz and even higher for the local areas allowed the spatial structure and dynamics of gas-vapor cavities thoroughly to be studied. The time dependence of the volume of the cavitation cavity on time was obtained for all investigated regimes. The vapor volume fraction, as expected, was found to increase with a decrease in cavitation number at higher flowrates. The feature of the separated cavitation cavity behind the cylinder is the complex structure consisting of the set of the flattened bubbles with a thin liquid interface between them. The collapse of the cavitation cavity proved to be always initiated only near one of the channel walls.

1. Introduction

Most attention is paid to cavitation in fluid machinery applications because it can produce noise, load variations and system vibrations and in total affect the performance. Ausoni et al. examines the effects of cavitation and fluid-structure interaction on the mechanism of the vortex generation and demonstrates a significant increase of the vortex-induced vibration level at cavitation onset \cite{1}. Another negative effect of cavitation is material erosion leading to fatigue failure of any hydraulic units. Understanding the cavitation processes is very important for prediction and control of its occurrence in the design of marine and hydraulic systems. The unsteady and unstable behavior of cavitation structures is still challenging for many researchers, and obtaining quality information about different cavitation systems is in demand.

Due to the impossibility of carrying out experiments on real hydraulic units various experimental and numerical studies have been performed on simpler geometries to reveal main flow and bluff body characteristics that lead to pressure drop and cavity formation. The characteristics of the cavitation around a bluff body depended on the body shape. Cavitation occurrence is usually associated with stagnation zones behind bodies and concentrated vortices in a wake. Among the main body types, the cylinder [2,3] with different aspect ratios can be distinguished.

The flow around a circular cylinder has been the subject of various studies in literature for both cavitating and non-cavitating conditions as it has broad engineering applications. The flow around the cylinder can be classified according to Reynolds number in sub-critical, critical and super-critical...
regimes. The upper subcritical range from $4\cdot 10^{4}$ is most interesting due to the lack of experimental data. Besides Reynolds number, the aspect ratio ($h/D$) is also a key parameter for flow patterns in a wake behind the body. The influence of aspect ratio on vortex shedding frequency and vorticity dilatation. Therefore, a circular cylinder is widely used to tune different numerical methods and revealed some flow features inaccessible from the experiment [5–7].

To further develop and verify simulation models describing cavitation, obtaining high-quality experimental information about the cavitation flow is necessary.

Due to the complex structure of the cavitation flow, its unsteadiness and spatial inhomogeneity, high-speed visualization is one of the most effective tools for studying fast processes [8,9]. The purpose of the current work is to cover the lack of experimental data for critically low aspect ratio when 3D complex cavitating flow structure becomes quasi-2D. In this paper, the cavitating flow around a cylindrical bluff body with aspect ratio $h/D=0.06$ is studied experimentally for different velocities of the incoming flow. It should be noted that the cylinder height to diameter ratio in the current study ($H/D = 0.06$) is considerably lower than the one used in work [4].

2. Experimental setup and technique

The experiments were carried out at the cavitation loop in the Institute of Thermophysics SB RAS with a 200 mm test section of constant 1.2×120 mm throat cross-section. A detailed description of the experimental setup can be found in work [10], with another installed test section reproducing the hydraulic turbine model. The upstream length is more than 50D, in order to have a fully-developed turbulent flow entering the test section. As a test body, the cylinder body with aspect ratio $h/D=0.06$ was chosen. The cylinder was made by machining one of the walls of the working section made of transparent acrylic, that provided no more than Ra 2.5 μm surface roughness. Average velocities calculated from flowrate were varied in the range from 5 to 20 m/s, but cavitation conditions occur starting from 14 m/s. To minimize the amount of dissolved gas, we used an iterative degassing method during several hours including turbulence of the flow at high Re, followed by evacuation of the evolved air bubbles with a vacuum pump from a reservoir located in the upper part of the experimental setup. The Reynolds number is calculated as $Re = V\cdot D_h/ν$, $ν$ - kinematic viscosity, $Q$ – flowrate, $D_h = 4S/P$ is the hydraulic diameter for a rectangular channel, where $S$ is the cross-sectional area, and $P$ is the wetted perimeter. The Strouhal number is calculated $Sh = f\cdot D/V$, $f$ – shedding vortex frequency, $D$ – body diameter, $V$ – bulk velocity. The cavitation cavity dynamics are captured using high-speed imaging by Photron fastcam nova s12 camera.

In order to study spatial structure and dynamics of gas-vapor cavities thoroughly, high-speed imaging at a sampling rate up to 200 kHz and even higher (for the local areas) was performed. A homogeneous light source, a LED panel, is placed behind the measurement section. This light source illuminates the area around the cylinder from the back side in the direction of the high-speed camera sensor.

With the selected lighting scheme, due to the scattering of light at the interface between vapor and water, the obtained images have a uniform light background with darker areas corresponding to the boundaries of cavities. Initially, the contrast of each image was increased. One percent of the lower and upper brightness values of the pixels were discarded, and the remaining values were rescaled to a range of 0-255. The boundaries of cavities were defined via binarization. The binarization threshold was calculated according to the Otsu method for the halftone image. That algorithm allows splitting the pixels into two classes – the cavity and background calculating a threshold that the intraclass dispersion was minimal. Assuming that the cavitation cavity occupies the entire space between the plates due to the small gap between them, the cavity volume for each image was calculated using pixel sum and channel height multiplication, which after calibration was converted to cubic millimeters.
Thus, we obtained the time dependence of the volume of the cavitation cavity on time for the investigated regimes (figure 3a).

3. Results
The investigation of the cavitation flow behind the circular cylinder with an aspect ratio of 0.06 at various incoming flow velocities was carried out. The Reynolds number is varied in the range from 30 000 to 45 000. The scenario of the formation of a cavitation cavity behind the body is similar for the same for all studied flow regimes (figure 1). Alternately, with a fairly constant frequency, a cavitation cavity appears on the left and right sides of the cylinder, increases in volume and then collapses. Figure 1 shows images with different configurations of the cavitation cavity obtained by an increase of the inlet flow rate from $Q = 7.3 \text{ m}^3/\text{h}$ to $Q = 8.3 \text{ m}^3/\text{h}$ which corresponds to $Re = 37500$ and $Re = 44200$. The maximum volume cavitation cavities are shown in figure 1 for each studied flow regime. The vapor volume fraction as expected was found to increase with a decrease in cavitation number at higher flowrates. To evaluate the cavitation intensity, the cavitation inception number is used:

$$\sigma = 2 \cdot \frac{(P_l - P_s)}{\rho V^2}$$

where $P_l$ - static pressure, $P_s$ - saturation pressure at current temperature, $\rho$ - water density, $V$ - bulk velocity. The hydraulic test rig has the system of a thermal stabilization that maintains a constant temperature, as a result, the saturation ware pressure is constant too. The cavitation number in the experiments was varied from 1.05 ($Q = 7.3 \text{ m}^3/\text{h}$) to 0.8 ($8.3 \text{ m}^3/\text{h}$).

![Figure 1. Maximum cavitation volume for different flowrates.](image)

Figure 2 shows the evolution of a typical cavitation cavity formed behind a circular cylinder with low aspect ratio at an incident flow velocity of about 18 m/s ($Q = 8.1 \text{ m}^3/\text{h}$). The time delay between frames is 0.3 ms.

The arising of a vapor cavity occurs near a circular cylinder in the region of its maximum expansion owing to significant overlap of the main flow in this region and, as a consequence, an increase local of velocity and decrease in pressure. Further, the attached cavitation cavity grows downstream. The cavity is transparent and homogeneous. The cavity spreads behind the cylinder passing into the central region of the channel. At the same time, a collapse of the elongated cavitation cavity attached to the cylinder is observed, and a regime with a cavitation region located far from the cylinder is realized. This region has some features that were not observed in earlier work [4] and which are realized due to the very small aspect ratio $h/D < 1$. 

3
The vapor cavity behind the circular cylinder consists of some flattened cavitation cavities. They are located close to each other and not in contact. These cavities can both narrow and expand. When small cavitation cavities expand, a single cavitation cloud with clearly defined boundaries of individual bubbles appears (figure 2).

One of the most important features in the collapse of a flattened cavitation cavity behind the body is that the beginning of the collapse does not occur simultaneously from two walls, as has been repeatedly shown for a circular cylinder, as well as other flow bodies, for example the hydrofoils [1,8,11] symmetrically from the channel walls namely the re-entrant jet forms at the walls of the working section concurrently, gradually spreading to the central area. In 2013 Decaix and Goncalvès [12] showed on the basis of numerical methods of mathematical modeling that there are two ways to realize the collapse of the attached cavitation cavity. The first is described above, the second one is characterized by alternating separation of cavitation clouds either near the left wall of the channel, then near the right. However, interleaving in objects of big scale is quite rare and its presence was experimentally demonstrated only in the work of the authors Timoshevskiy et al. [9]. When considering the collapse of the cavity on objects with a small aspect ratio, only this collapse method is observed. After the collapse, a conglomerate of microbubbles is formed, which are carried downstream and dissolve in water with an increase in local pressure.

![Figure 2. The full cycle of cavitation cavity formation on left and right sides.](image)

Based on the temporal realization of the cavitation cavity volume for each mode, spectrograms were calculated using fast Fourier transform (figure 3b).

![Figure 3. The volume fluctuation of cavitation area and spectrogram at Q = 8 m³/h.](image)
One can see in figure 3b, the main frequency of the formation of the cavitation cavity behind the circular cylinder with a very small aspect ratio for \( Q = 8 \text{ m}^3/\text{h} \) is 217 Hz. Figure 4 demonstrates the generalizing dependence of the frequency of formation of cavitation cavities on the flow rate, which is a weakly nonlinear.

![Figure 4](image_url)

**Figure 4.** Dependence of the separation frequency of the cavitation cavities on the flow rate.

### 4. Conclusion

The obtained frequency of the formation of cavitation cavities is in the range 200-235 Hz for studied flow regimes. It was found that the Strouhal number for the obtained frequency range corresponds to approximately 0.38, which is significantly higher than the values of \( Sh \) close to 0.2 found in the literature [3] and requires further research and explanation. The separated cavitation cavity behind the cylinder has the multiple structures basically. The cavity is formed from a set of the flatted bubbles with a liquid interface between each other in most cases. The collapse of the cavity is always initiated near one of the channel walls. The obtained experimental data, taking into account the canonicity of the experiment, contribute to the verification of numerical codes and analytical models in the description of cavitation phenomena. The work is the beginning of a cycle of research in the transition to microscale heights of flow bodies in order to get insights into cavitation physics. Further, it is planned to study a more realistic model of the NACÂ geometry.

### Acknowledgments

The study was supported by the grant of the Russian Science Foundation (Project No. 19-79-10217).

### References

[1] Ausoni P, Farhat M, Escaler X, Egusquiza E and Avellan F 2007 Cavitation influence on von Kármán vortex shedding and induced hydrofoil vibrations *J. Fluids Eng. Trans. ASME* **129** 966–73

[2] Kumar P, Bakshi S and Chatterjee D 2017 Experimental investigation of cavitation behind a circular cylinder in cross-flow *J. Therm. Sci. Eng. Appl.* **9**

[3] Asano Y, Watanabe H and Noguchi H 2020 Effects of cavitation on Kármán vortex behind circular-cylinder arrays: A molecular dynamics study *J. Chem. Phys.* **152**

[4] Gonçalves R T, Franzini G R, Rosetti G F, Meneghini J R and Fujarra A L C 2015 Flow around circular cylinders with very low aspect ratio *J. Fluids Struct.* **54** 122–41

[5] Brandao F L, Bhatt M and Mahesh K 2019 Numerical study of cavitation regimes in flow over a circular cylinder *J. Fluid Mech.*

[6] Gnanaskandan A and Mahesh K 2016 Numerical investigation of near-wake characteristics of
cavitating flow over a circular cylinder [*J. Fluid Mech.* **790** 453–91]

[7] Ghahramani E, Jahangir S, Neuhauser M, Bourgeois S, Poelma C and Bensow R E 2020 Experimental and numerical study of cavitating flow around a surface mounted semi-circular cylinder [*Int. J. Multiph. Flow* **124** 103191]

[8] Kravtsova A Y, Markovich D M, Pervunin K S and Timoshkevskyi M V. 2013 High-speed imaging and PIV measurements in turbulent cavitating flows around 2D hydrofoils [*Int. Symp. Turbul. Shear Flow Phenomena, TSFP 2013*]

[9] Timoshkevskyii M V., Churkin S A, Kravtsova A Y, Pervunin K S, Markovich D M and Hanjalić K 2016 Cavitating flow around a scaled-down model of guide vanes of a high-pressure turbine [*Int. J. Multiph. Flow* **78** 75–87]

[10] Skripkin S, Tsoy M, Kuibin P and Shtork S 2019 Swirling flow in a hydraulic turbine discharge cone at different speeds and discharge conditions [*Exp. Therm. Fluid Sci.* **100**]

[11] Huang B, Ducoin A and Young Y L 2013 Physical and numerical investigation of cavitating flows around a pitching hydrofoil [*Phys. Fluids* **25**]

[12] Decaix J and Goncalves E 2013 Investigation of three-dimensional effects on a cavitating Venturi flow [*Int. J. Heat Fluid Flow* **44** 576–95]