Applying PIV to study a fluid flow in the vicinity of a circular streamlined cylinder

K G Dobroselsky
Kutateladze Institute of Thermophysics, 1 Ac. Lavrentiev ave., Novosibirsk, Russia
E-mail: dobroselsky@mail.ru

Abstract. The round cylinder streamlining by a turbulent water flow in a hydrodynamic pipe is investigated experimentally. Using the optical method of PIV (Particle Image Velocimetry) the averaged velocity fields near the cylinder are obtained and, on their basis, the geometric characteristics of the vortex zone of the near wake are calculated for the non-cavitation and cavitation flow regimes in the critical region of Reynolds numbers. The use of two mirrors, set at a certain angle to each other, allows obtaining a picture of the velocity fields around the entire cylinder, rather than around half of it, as it is done in most of the works. Using vector patterns of averaged velocity fields, the angles of the boundary layer separation from the cylinder surface along the reverse flow are determined in the considered flow regimes. An asymmetrical separation of the boundary layer from different sides of the streamlined cylinder was obtained. It is shown that the increase in the Reynolds number for the non-cavitation flow regimes leads to a more than two times decrease in the vortex zone behind the cylinder, and, accordingly, to a displacement of the separation angles downstream. Cavitation increases the vortex zone behind the cylinder and displaces the separation angles upstream.

1. Introduction

Cross-streamlined cylindrical surfaces are wide spread in heat exchange, energy and thermal engineering devices. Despite the simple geometry, the flow around the circular cylinder is extremely complex and depends on the flow regime.

Analysis of the literature shows that most of the works on the round cylinder streamlining are associated with the flow of gas and, less often with liquid. There are practically no experimental works on the cavitation flow around the cylinder [1, 2]. Cavitation, in general, is undesirable in many designs, so one needs to know the boundaries of the parameters and the conditions under which it occurs and affects the necessary characteristics.

Conditions of streamlining and the size of the vortex zone (structures) behind the bodies are important for designing the arrangement of both single and groups of streamlined elements in hydraulic and thermal structures. It should be also noted that for building and verifying modern mathematical models, describing the emergence and development of cavitation, it is extremely important to obtain experimental information for cavitation flow regimes of model objects.

This paper, using the PIV method, experimentally studies the velocity field near a round steel cylinder, streamlined by a turbulent water flow, in the critical region with Reynolds numbers over the diameter Re ≈ (2.07-2.84)×10^5 for the non-cavitation and cavitation regimes. Methods for determining the average spatial characteristics of the near vortex wake behind the cylinder are shown.
2. Experimental methodology
The hydrodynamic tube of closed type of Novosibirsk National Research State University was used for experimental studies. Measurements of velocity fields were realized using the PIV system POLIS, developed at the Institute of Thermophysics SB RAS and consisting of: a programmable sync processor; dual solid-state pulse Nd:YAG laser with a wavelength of the laser radiation of 532 nm (pulse energy of 25 mJ, pulse duration of 10 ns, and pulse repetition rate up to 1.3 Hz) with focusing and cylindrical lenses to create a laser sheet; a CCD camera with a resolution of 2048×2048 pixels [3].

Figure 1 shows the main elements of the POLIS and the arrangement of the cylinder in the working area. Perpendicular to the side walls of a rectangular channel with a cross section of 0.08×0.15 m (width × height) there was a round steel cylinder with a diameter \( d = 0.026 \) m. The walls of the working area were equipped with viewing windows for optical measurements. The cylinder was streamlined by the water flow with a constant volumetric flow rate. The ratio of the flow rate to the cross-sectional area of the working section served to determine the superficial velocity \( u \) of the incoming flow. The flow rate was measured using an ultrasonic flowmeter with a relative volume error of no more than 2%. Pressure and temperature before and after the cylinder were measured by two sensors of pressure and resistance thermometers, respectively, with an error of no more than 0.5% [3]. The working fluid was filtered tap water. All experiments used polyamide tracers with an average size of 50 μm and a density close to that of water.

![Figure 1. Experimental scheme: 1 – flow, 2 – mirror, 3 – test section, 4 – CCD camera, 5 – mirror, 6 – laser; white points – tracers.](image)

Control over the experiment and data processing were carried out using the "ActualFlow" software package [4], designed for automating the experimental process, data processing and visualization. The thickness of the laser "sheet" formed by a cylindrical lens was about 1 mm in the measuring area of the central longitudinal section of the working channel. The size of the measuring area was approximately 125×125 mm.

Instantaneous velocity fields were calculated using an iterative cross-correlation algorithm with continuous displacement and deformation of elementary computational cells and 75% overlap of computational domains. The sub-pixel interpolation of the cross-correlation peak was realized on three points, using a one-dimensional approximation by the Gauss function. The initial size of the elementary computational domain was 64×64 pix, and the final size was 8×8 pix. The obtained vector
fields of instantaneous velocity underwent two validations: by signal/noise ratio with a factor of 2 and by an additive median filter in a region of 7×7 pix. The velocity measurement error depends on the tracer displacement and the number of frames; in our case it was on average 2% and 5% at the displacement by 8 and 2 pixels, respectively [5].

In advance, the velocity field was measured by PIV method in the vertical central section (1000 double images) for various flow rates in a free working area. The velocity profile was found to have a core of constant velocity (the boundary layer on the channel walls was 10-12 mm) with a level of turbulence of the longitudinal velocity component of the order of 0.8-1%.

3. Results and Discussion

Figures 2 and 3 show the fields of averaged velocities \( v_x \) and \( v_y \) for the non-cavitation regime of the cylinder streamlining in the central part of the channel at a superficial velocity \( u = 8.8 \) m/s, the velocity in the free flow core \( v_{\text{max}} = 9.8 \) m/s, the water temperature \( t \approx 25 \) °C and the corresponding Reynolds number \( Re \approx 2.84 \times 10^5 \). Averaging was carried out over 2000 double images. Here \( x, y \) are the longitudinal and transverse coordinates related to diameter \( d \), respectively. Figure 3 shows the area with half of a cylinder: \( x = 0-1.07; y = -0.54 \pm 0.54 \).

![Figure 2. Average velocity field \( v_x \) for non-cavitation regime of cylinder streamlining.](image)

![Figure 3. Determination of the main parameters of the near wake behind the cylinder (velocity field \( v_y \)).](image)

Just behind the cylinder, a stable turbulent region forms. There opposite rotating vortices appear: one rotates clockwise (from the top), and the second one rotates counterclockwise (from the bottom). Between the vortices, the liquid moves in the direction opposite to the main flow.

In figure 3, white horizontal and vertical dashed lines show the length of the reverse flow \( l \), and the distance between the vortices \( \Delta y \), respectively. The outer boundary of the region of reverse flows is limited by a line, on which the longitudinal velocity component is zero.

On vector pictures of the averaged velocity field, it is possible to determine the final separation angle \( \varphi \) (relative to the frontal point) of the boundary layer along the boundary of the return motion on the cylinder surface. Figure 3 shows the location of the separation angles \( \alpha_+ \) and \( \alpha_- \) with respect to the upper and lower surfaces of the cylinder and from the rear point, where \( \alpha_+ = \tan^{-1}(\Delta y/\Delta x) \) and \( \alpha_- = \tan^{-1}(\Delta y/\Delta x) \). The final separation angles are determined as \( \varphi_+ = 180^\circ - \alpha_+ \) and \( \varphi_- = 180^\circ - \alpha_- \), respectively. Figure 4 (a-c) shows the average (2000 double images) fields of relative velocities \( v_x/u \) at a non-cavitation flow around the cylinder in the central part of the channel for different Reynolds numbers.
Figure 4. Average velocity field $v/u$: non-cavitation regimes (a) – $Re = 2.07 \times 10^5$, (b) – $Re = 2.54 \times 10^5$, (c) – $Re = 2.84 \times 10^5$; cavitation regime (d) – $Re = 2.84 \times 10^5$.

The streamlining at the cavitation regime (figure 4 (d)) occurred at the relative cavitation number $\sigma/\sigma_i \approx 0.74$, where $\sigma_i$ corresponds to the beginning of cavitation. Figure 4 (c) and (d) show comparative pictures of velocity fields at the same Reynolds number, the same temperature, but different pressures before the cylinder, $p = 161$ kPa and 112 kPa, respectively.

Below, figure 5 presents comparative graphs of $-\tau q$ from $y$ for different $Re$, where $\tau = -\rho u_x u_y$ is the turbulent stress ($u_x, u_y$ – corresponding pulsations of velocities $v_x$ and $v_y$; $\rho$ – water density); and $q = \rho u^2/2$ is the average dynamic head of the incoming flow. According to the graphs, it is possible to estimate the transverse size of the vortex zone of the near wake ($\Delta y_w$) behind the cylinder, for example, where the solid line comes to zero above and below.
Figure 5. Profiles of relative turbulent stresses: (a) – Re = 2.07×10⁵, (b) – Re = 2.54×10⁵, (c) – Re = 2.84×10⁵, (d) – Re = 2.84×10⁵ (cavitation regime). The solid line corresponds to the coordinate x, at which the velocity of the reverse flow behind the cylinder is maximal, the dotted line – x = 1.2, and the dashed line – x = 1.5.

The table 1 below shows the main spatial parameters of the vortex zone behind the cylinder for non-cavitation regimes.

| Re (10⁵) | 2.07 | 2.37 | 2.54 | 2.84 |
|----------|------|------|------|------|
| Angles of boundary layer separation $\varphi_+ , \varphi_-$ | 105°, 113° | 115°, 125° | 131°, 140° | 134°, 141° |
| Length of reverse flow $l_r (d)$ | 1.00 | 0.72 | 0.60 | 0.45 |
| Width of vortex zone $\Delta y_w (d)$ | 1.6 | 1.0 | 0.8 | 0.7 |
For cavitation streamlining of the cylinder, the flow parameters were as follows: the separation angles were \( \phi_+ = 129^\circ \), \( \phi_- = 138^\circ \); \( l_r \approx 0.65d \); \( \Delta y_w \approx 1.0d \). In [6, 7] for \( \text{Re} = (1-2) \times 10^5 \) there are given \( l_r \approx (0.7-1.05)d \).

The asymmetry of the separation of the boundary layer from the surface of the cylinder is associated with the formation of a laminar separation bubble, which at critical Reynolds numbers (\( \text{Re} \approx (2-4) \times 10^5 \)) forms on one side. In this regime, transition to turbulence first occurs in one of the boundary layers and it characterized by the separation with further reattachment of the boundary layer, forming a bubble [8]. The final separation of a turbulent boundary layer occurs at \( \phi \approx 140-147^\circ \) [8, 9].

The asymmetry and displacement of the near wake upwards can be related to the pressure difference in the flow at the level of the upper and lower parts of the cylinder.

4. Conclusions

The averaged velocity fields obtained by the PIV method allow a reliable investigation of the flow in the vicinity of the streamlined cylinder. It is shown that the kinematic characteristics of the wake behind the cylinder depend on the value of the Reynolds number, as well as on the flow regime.

It has been found that for the non-cavitation flow regimes, with an increase in the Reynolds number, the vortex zone of the near wake behind the cylinder decreases more than two times, and the separation angles of the boundary layer increase by 25%. The vortex zone behind the cylinder is found to increase both in the transverse and longitudinal direction in the cavitation flow regime, relative to the non-cavitation flow.

Acknowledgments

The work was financially supported by the Russian Science Foundation (grant No.19-79-30075).

References

[1] Ihara A and Murai H 1986 Cavitation inception on a circular cylinder at critical and supercritical flow range Journal of Fluids Engineering 108 421
[2] Kumar P, Chatterjee D and Bakshi S 2017 Experimental investigation of cavitating structures in the near wake of a cylinder International Journal of Multiphase Flow 89 207
[3] Dobrosel'skii K G 2016 Flow around a cylinder at the beginning of the critical region Journal of Applied Mechanics and Technical Physics 57 No 2 293
[4] Akhmetbekov E K, Bil'skii A V, Lozhkin Yu A, Markovich D M and Tokarev M P 2006 System of control of experiment and processing of data obtained by the methods of digital tracer visualization (ActualFlow) Vych. Metod. Programmirov. 7 79
[5] Dobroselsky K G 2018 Studying the flow around a cylinder the critical region by the PIV method J. Phys.: Conf. Series 1105 012014
[6] Djeridi H, Braza M, Perrin R, Harran G, Cid E and Cazin S 2003 Near-wake turbulence properties around a circular cylinder at high Reynolds number Flow, Turbulence and Combustion 71 19
[7] Perrin R, Cid E, Cazin S, Sevrain A, Braza M, Moradei F and Harran G 2007 Phase-averaged measurements of the turbulence properties in the near wake of a circular cylinder at high Reynolds number by 2C-PIV and 3C-PIV Exp. Fluids 42 93
[8] Lehmkuhl O, Rodriguez I, Borrell R, Chiva J and Oliva A 2014 Unsteady forces on a critical Reynolds number Phys. Fluids 26 125110
[9] Marr Yu N and Shvegzhda S A 1989 Features of fluid motion near the surface of the cylinder in transverse flow Izv. Akad.Nauk SSSR, Mekh. Zhidk. Gaza 4 65