 Numerical study of the flow around a circular cylinder with the slip length boundary effect at a critical Reynolds number

A. Sentyabov1,2, A. Gavrilov1,2, and A. Dekterev1,2
1Kutateladze Institute of Thermophysics SB RAS, Novosibirsk, Russia
2Siberian Federal University, Krasnoyarsk, Russia
Sentyabov_a_v@mail.ru

Abstract. The paper presents an investigation of the slip length effect on the flow around a circular cylinder at Reynolds number \( Re = 2.5 \times 10^5 \). The study was performed by means of numerical simulation of the flow with the URANS approach based on the \( k-\omega \) SST model. Calculations show a significant effect of the slip length on the flow patterns. With an increase in the slip length, the drag coefficient noticeably decreases and the pulsations of the lift force reduce. With an increase in the slip length, the separation of the flow from the cylinder is delayed, which significantly affects the flow patterns in the wake behind the cylinder.

1. Introduction
The flow around a circular cylinder is one of the simplest examples of external flow, demonstrating a wide variety of flow patterns and related phenomena, such as flow separation and the formation of a vortex street. For this reason, this flow is a good test for practicing various flow control methods, one of which is the microstructuring of the hydrophobic surface, which allows water to "slip" over the surface holding air bubbles in the cavities [1]. To study the effect of the properties of such a surface on the flow around a circular cylinder, we carried out a numerical simulation of the flow with the boundary conditions using slip length that simulate the properties of a hydrophobic surface.

2. Numerical model
The URANS method based on the \( k-\omega \) SST model [2] is used to simulate turbulent flows. The used numerical algorithm is based on the finite volume method for an unstructured mesh. The flow around a body with a hydrophobic surface is modeled using special boundary conditions with slip length, at which there is a velocity step on the wall proportional to the corresponding gradient:

\[
\nu_l - \nu_w = b \frac{\partial \nu}{\partial n}
\]

where \( \nu_l \) is the velocity of the liquid on the wall, \( \nu_w \) is the velocity of the wall, \( b \) is a coefficient called the slip length. These conditions were implemented iteratively by means of Dirichlet boundary conditions.

The object of the research was a three-dimensional flow around a single cylinder in the channel of a hydraulic stand (figure 1). A circular cylinder with diameter \( D = 26 \text{ mm} \) was confined in the channel 80 mm wide (3.08\( D \)), 150 mm high (5.77\( D \)) and 1000 mm long (38.5\( D \)). The distance from the
entrance to the axis of the cylinder was 500 mm (19.2D), and the length of the cylinder was 80 mm (3.08D). At the inlet uniform distribution of the longitudinal velocity \( U = 9.8 \text{ m/s} \) was set. A fixed absolute pressure was set at the outlet. The no-slip conditions were set on the side, upper and lower boundaries. Water was used as a fluid. The density was considered equal to \( \rho = 1000 \text{ kg/m}^3 \), and the viscosity was \( \mu = 0.001 \text{ Pa}\cdot\text{s} \). Thus, the Reynolds number was \( 2.5 \cdot 10^5 \).

![Figure 1. Scheme of the calculated domain.](image)

The block-structured computational mesh was used to simulate the flow. Parameters of the mesh were as follows: 160 cells along the circumference of the cylinder with condensation in the wake area 80 cells along the cylinder length. The mesh was thickened towards the cylinder wall and side walls with \( y^+ < 2.5 \) and \( y^+ < 3.5 \), respectively. In total, the computational mesh contained 1.69 million hexahedral cells. The time step was 0.0001 s or \( 0.04D/U \), which corresponds to the Courant criterion \( C < 6 \) in the entire domain. The computational mesh is shown in figure 2. A preliminary nonstationary calculation was carried out for 0.5 s, which is about \( 185D/U \), during which a periodic vortex shedding developed. Then, averaging was carried out over \( 130D/U \) (0.35 s).

![Figure 2. An example of the mesh.](image)

### 3. Results
Calculations with different meshes up to \( 8 \cdot 10^6 \) cells were performed for validation. Validation showed that: 1) the drag coefficient for meshes with a fine resolution along the length of the cylinder fits within the experimental data [3] and the results of different meshes differ within 10%; 2) the pulsations of the lift coefficient for the same meshes get into the range of experimental data [4]. Thus, the calculated data got into the range of the experimental data.

Mean values for the calculations with various slip length are summarized in Table 1, which shows:

- \( C_D = F_x/(0.5 \rho U^2 D) \) is the mean drag coefficient,
- \( L_r/D \) is the respective recirculation length,
- \( St = fD/U \) is the Strouhal number
$\Delta C_L$ is the rms of the lift coefficient pulsations.
$V_{\text{maxr}}$ is maximal mean reverse flow
$V_{\text{maxwall}}$ is maximal velocity magnitude on the cylinder surface
($F_x$ is the drag force, $f$ is the vortex shedding frequency)

Calculations in a wide range of slip lengths show a significant effect of the properties of the cylinder surface on the flow patterns. With an increase in the slipping length, the drag coefficient rapidly decreases by almost two times (Table 1, figure 3). Slip length has an even greater effect on the lift force pulsations which are reduced by more than an order of magnitude. As figure 4 shows, with an increase in the slip length, the separation of the flow is delayed, which significantly affects the flow in the wake. First, the recirculation zone is stretched and narrowed. With a further increase in the slip length, the separation point noticeably shifts downstream, and the recirculation zone decreases again (Figure 5). At $b = 0.01$ mm, the length of the recirculation zone and the intensity of the reverse flow increase. At $b = 0.05$ mm, the length of the recirculation zone increases, but the reverse flow velocity remains the same. At $b = 0.1$ mm, the length decreases. At $b = 0.15$ mm, the size and intensity of the recirculation zone decrease.

Table 1. Results of calculations

| $b$ (mm) | $b/D$ (%) | $C_D$  | $L/D$ | St   | $\Delta C_L$ | $V_{\text{maxr}}/U_0$ | $V_{\text{maxwall}}$ (m/s) |
|----------|-----------|--------|-------|-------|--------------|------------------------|-----------------------------|
| 0.15     | 0.577     | 0.383  | 0.65  | 0.149 | 0.01         | 0.24                   | 17.2                        |
| 0.10     | 0.385     | 0.390  | 0.92  | 0.1   | 0.011        | 0.39                   | 15.6                        |
| 0.05     | 0.192     | 0.662  | 1.07  | 0.308 | 0.128        | 0.3                    | 12.3                        |
| 0.01     | 0.038     | 0.760  | 0.88  | 0.292 | 0.243        | 0.31                   | 9.0                         |
| 0        | 0.000     | 0.856  | 0.65  | 0.287 | 0.445        | 0.29                   | 0                           |

Figure 3. Drag coefficient ($C_D$) and rms of the lift coefficient pulsations ($\Delta C_L$) versus slip length.
Figure 4. Mean longitudinal velocity behind the cylinder versus slip length.
Figure 5. Mean longitudinal velocity behind the cylinder along the channel axis versus slip length.

4. Conclusions
Calculations in a wide range of slip lengths show a significant effect of the properties of the cylinder surface on the flow patterns. With an increase in the slip length, the drag coefficient rapidly decreases by almost two times. Slip has an even greater effect on the pulsations of the lift force, which are reduced by more than an order of magnitude. In fact, the formation of the Karman vortex street is suppressed. With an increase in the slip length, the separation of the flow from the cylinder is delayed, which significantly affects the flow in the wake. First, the recirculation zone stretches by about 30% and narrows. With a further increase in the slip length, the separation point noticeably shifts downstream, and the recirculation zone length decreases again.

Acknowledgments
The research was supported by a grant from the RF government aimed at the governmental support of research handled by the leading scientists in Russian universities (leading scientist C. Markides) (grant # 075-15-2019-1888).

References
[1] Muralidhar P, Ferrer N, Daniello R, Rothstein J P 2011 J. Fluid Mech. 680 459–476.
[2] Menter F R 1994 AIAA Journal 32(8) 1598-1605.
[3] Fung Y 1960 Journal of the aerospace science 27(11) 801–814.
[4] Norberg C 2003 Journal of Fluids and Structures 17(1) 57–96.