DNS of heat transfer of the flow over a cylinder
at $Re = 200$ and 1000

E Palkin$^{1,}$, V Ryzhenkov$^{1,2}$ and R Mullyadzhanov$^{1,2}$

$^1$Institute of Thermophysics SB RAS, 630090, Novosibirsk, Russia
$^2$Novosibirsk State University, 630090, Novosibirsk, Russia

E-mail: palkinev89@gmail.com

Abstract. We perform Direct numerical simulations of the flow over a cylinder to study the influence of the thermal boundary conditions on the Nusselt number distribution. The results show very good agreement with some data from the literature. The total Nusselt number for the constant heat flux (CHF) boundary condition is higher for some 15% than for the constant wall temperature (CWT) condition for both $Re < 200$ and 1000.

1. Introduction
Heat transfer and forced convection of the flow over a cylinder has been the subject of many numerical and experimental studies representing a canonical bluff body configuration. The drag and lift forces acting on the body due to periodic formation and shedding of typical large-scale vortices lead to flow-induced vibrations and can cause serious structural damages. Effective cost-beneficial control methods of flow-induced drag and lift reduction and heat transfer enhancement require prior understanding of the flow and its vortical dynamics. This case featuring multiple regimes depending on the Reynolds number $Re$ has been thoroughly studied by many authors [1, 2]. For $Re < 150$ the flow is steady and two-dimensional, in the range $150 < Re < 300$ the laminar transitional regime is observed and at $300 < Re < 10000$ the wake becomes irregular and three-dimensional with laminar-turbulent transition occurring in the separated shear layer.

An important issue concerning the heat transfer is the influence of thermal boundary conditions on the wall which usually comes down to the constant wall temperature (CWT) or constant heat flux (CHF) condition. There are a number of extensive reviews about the cylinder heat transfer focusing on various effects [3, 4]. Despite a large amount of studies on this flow configuration, even for relatively low Reynolds numbers, the existing data are significantly scattered. It can partially be explained by differences in parameters like freestream turbulence level, blockage and duct channel aspect ratio, boundary conditions on the cylinder or flow features like low-frequency modulations in irregular flow regime [5-7], which affect both the flow dynamics and heat transfer from the wall. Boulos & Pei [8] reported that the total Nusselt number differs within 10-20% between CWT and CHF for $Re = 5300$. The difference between peak values of the wall heat flux for CWT and CHF was found to be around 50% at $Re = 80000$ reported by Papell [9] and confirmed by other authors [10]. The scatter is also present for higher $Re = 140000$ [11]. Widely accepted correlations are usually constructed based on the data with different parameters (aspect ratio, turbulence level etc) [12]. In the present paper we investigate the heat transfer with CHF and CWT boundary conditions by means of direct numerical simulations to analyze this issue.
2. Computational details

We perform Direct numerical simulations (DNS) of the uniform flow over a cylinder with a heated wall with a relatively fine spatial resolution. The Reynolds number corresponds to 200 and 1000 with the Prandtl number equal to 0.71 (air). For $Re = 200$ the flow is laminar with unsteady von Karman vortex street, while at $Re = 1000$ the flow is substantially three-dimensional and laminar-turbulence transition occurs in separated shear layer. To compare with data from the literature for $Re = 200$ we use a two-dimensional setup while for $Re = 1000$ the geometry is three-dimensional with the computational domain representing a box with $25D \times 20D \times 6D$ in $x \times y \times z$ directions as shown in Fig. 1. A circular cylinder with diameter $D$ is located at the origin $x = 0$, $y = 0$. The inflow boundary is taken at $x = -10D$, while the outflow is $x = 15D$ downstream. At the side boundaries $|y| = 10D$ zero normal velocity condition is imposed and periodic boundaries are set alongside the cylinder at $z = 0$ and $6D$ with $L = 6D$ being the length of the cylinder. For the temperature field either constant heat flux (CHF) or constant wall temperature (CWT) condition is used on the cylinder wall, where the Nusselt number can be defined representing a non-dimensional heat flux.

We use the computational code Nek5000 featuring semi-implicit third-order time integration scheme and spatial discretization based on the spectral-element method (SEM). The Navier–Stokes equations are discretized in space with the use of the Galerkin approximation with 8th-order Lagrange polynomial interpolants based on the Gauss–Lobatto–Legendre points for both the velocity and pressure fields ($P_N$–$P_N$ formulation). In circumferential direction alongside the cylinder we use 24 spectral elements for both Reynolds numbers and 12 spectral elements are used in $z$-direction for $Re = 1000$. In total the mesh consists of 0.25 and 16 mln points for $Re = 200$ and $Re = 1000$, respectively.

![Figure 1](image.jpg)

**Figure 1.** Computational domain, coordinate system (placed in the center of the cylinder) and isosurfaces of the instantaneous $Q$-criterion for $Re = 1000$ and CWT boundary condition colored with the non-dimensional temperature field $T$ in the range [0; 0.25].
3. Results

The surface of the instantaneous $Q$-criterion presented in Fig. 1 shows a turbulent three-dimensional flow behind the cylinder similar to experimental observations [2] with attached recirculation zone and periodic vortex shedding.

![Figure 2: Time history of the drag and lift coefficient $C_D$ = $F_x/(L \rho U^2/2)$, $C_L$ = $F_y/(L \rho U^2/2)$, where $F_x$ and $F_y$ are the total forces acting on the cylinder.](image)

The time history of the drag and lift coefficients is shown in Fig. 2 demonstrating a sinusoidal signal for $Re = 200$, while for $Re = 1000$ it is somehow modulated. The period of lift coefficient corresponds to the Strouhal number $St = 0.19$ for $Re = 200$ matching the results of Bouhairie & Chu [13]. For $Re = 1000$ the shedding period corresponds to $St = 0.219$ and this value is within a good agreement to other authors [1,13-15,18]. The mean drag coefficient $C_D = 1.34$ for $Re = 200$ while Piñol & Grau [16] found the value of 1.27. For $Re = 1000$ the drag coefficient is significantly lower with $C_D = 0.99$ since the recirculation zone is elongated, which falls in the scatter of the data from the literature [14,17-19].

![Figure 3: Circumferential pressure coefficient distribution along the cylinder surface $C_p = (p - p_s)/(D \rho U^2/2)$. Red line – data from Zhao et al. [17], green – Lei et al. [18].](image)

The pressure distribution along the wall is presented in Fig. 3 in comparison with the literature data [17, 18] for $Re = 1000$. As the separation point moves upstream with the increase of $Re$, the pressure minimum shifts from the angle $\theta = 80$ to 71 degrees assuming $\theta = 0$ is the frontal point. The difference between present and reference data is likely to be caused by the mesh resolution, which is 0.73 and 0.4 mln points in [17] and [18].
Figure 4. White streamlines correspond to the time-averaged velocity field together with a streamwise component contours for $Re = 200$ and $Re = 1000$. Black streamlines correspond to the data from Zhao et al. [17].

The time-averaged streamwise velocity field together with streamlines is shown in Fig. 4. The data from the literature [17, 18] feature a secondary recirculation zone attached to the cylinder surface, which is a footprint of the low numerical resolution [5, 7].

Figure 5. Circumferential time-averaged Nusselt number distribution on the cylinder wall corresponding to CWT/CHF normalized by the peak value. Left: $Re = 200$, $Nu_{max} = 12.65$ [13], 14.69 [20], 16.46/16.75 [21], 14.0 [22], 13.4 [23] and 13.41 with the present calculations. Right: $Re = 1000$, $Nu_{max} = 29.43$ [13], 31.3 [22], 30.0 [23] and 30.00 with the present calculations.

The Nusselt number circumferential distribution is plotted in Fig. 5 against the reference data [13, 20–21] normalized by the peak $Nu$ value. Nusselt profile for $Re = 200$ and CWT boundary condition is in rather good agreement with Bouhairie & Chu data, although the peak value is slightly higher. For $Re = 1000$, the Nusselt number at focal point is in agreement with the reference data, however after the separation point they start to diverge, which is likely to be caused by two-dimensional approach applied in [13]. The overall Nusselt number is 7.43/8.59 and 15.27/17.42 corresponding to CWT/CHF for $Re = 200$ and 1000, respectively, while reported value in [24] is 6.58 and 14.3 for $Re = 200$ and 1000 correspondingly.
Conclusion
We performed Direct numerical simulations of the flow over a cylinder to study the influence of the thermal boundary conditions on the distribution of the Nusselt number. The results are in good agreement with the data from the literature. However for $Re = 1000$ the reference data obtained by a two-dimensional approach [13] show some deviations after the separation point. Generally the total Nusselt number for the constant heat flux (CHF) boundary condition is higher for some 15% than for the constant wall temperature (CWT) condition for the considered cases.

Acknowledgements
This work is funded by the Russian Science Foundation, grant No. 19-79-30075. The computational resources are provided by Siberian Supercomputer Center SB RAS, Joint Supercomputer Center of the Russian Academy of Sciences and Supercomputing Center of the Novosibirsk State University.

References
[1] Roshko A 1954 NACA TN 3169
[2] Williamson C 1988 Phys. Fluids 31 3165–8
[3] Zukauskas A 1972 Adv. Heat Transf. 8 116–33
[4] Morgan V 1975 Adv. Heat Transf. 11 199–264
[5] Lehmkuhl O, Rodrı́guez I, Borrell R and Oliva A 2013 Phys. Fluids 25 (8) 085109
[6] Peng S, Wang H, Zeng L and He X 2019 Exp. Therm. Fluid Sci. 109 109877
[7] Palkin E, Mullyadzhanov R, Hadžiabdić M and Hanjalić K 2016 Flow Turbul. Comb. 97 1017–46
[8] Boulos M and Pei D 1974 Int. J. Heat Mass Transf. 17 (7) 767–83
[9] Papell S 1981 NASA report
[10] Baughn J and Saniei N 1991 J. Heat Transf. 113 (4)
[11] Hadžiabdić M, Palkin E, Mullyadzhanov R and Hanjalić K 2019 Int. J. Heat Fluid Flow 79 108441
[12] Churchill S and Bernstein M 1977 J. Heat Transf. 99 (2) 300–6
[13] Bouhaire S and Chu H 2007 J. Fluid Mech. 570 177–215
[14] Mittal S 2001 Phys. Fluids 13 177–91
[15] Norberg C 1994 J. Fluid Mech. 258 287
[16] Piñol S and Grau F 1998 Numer. Heat Transf., Part A: Applic. 34 313–30
[17] Zhao M, Cheng L and Zhou T 2009 J. Fluids Struct. 25 831–47
[18] Lei C, Cheng L and Kavanagh K 2001 Computer Methods in Appl. Mech. and Engineer. 190 2909–23
[19] Szeczenyi E 1975 J. Fluid Mech. 70 529–42
[20] Patnaik B, Narayana P and Seetharamu K 1999 Int. J. Heat Mass Transf. 42 (18) 3495–507
[21] Chun W and Boehm R 1989 Num. Heat Transf. 151(1) 101-122
[22] Squire H 1938 Modern Developm. Fluid Dyn. 631–2
[23] Frossling N 1958 Tech. Rep. 1432
[24] Eckert E and Drake R 1972 Analysis Heat Mass Transfer