Numerical Simulation of 3D Flow over a Circular Cylinder

A N Kusyumov, S A Kusyumov, S A Mikhailov and E V Romanova
Tupolev Kazan National Research Technical University, Kazan, 420111, Russia
E-mail: postbox7@mail.ru

Abstract. Unsteady 3D flow over a circular cylinder at Reynolds number of 3900 is studied numerically using the Navier-Stokes equations. Two formulations of the problem were considered: with boundary conditions corresponding to the flow around an isolated cylinder and with periodic boundary conditions to the flow behind a parallel circular cylinders grid. A comparative analysis of the integral and distributed characteristics of the flow around the cylinder and the spectral characteristics of the flow for both formulations of the problem is carried out.

1. Introduction
Flow over a cylinder has been studied both experimentally and numerically as referred in the reviews [1], [2]. Simulation of flow over circular cylinders is a simple model for different applications a concerning study of flow around bluff bodies. This task encapsulates all typical flow peculiarities inherent areas of turbulence generation. These areas include the boundary layer on the cylinder surface, the two shear layers delimiting the recirculation region, and the vortex wake [3, 4]. Despite geometric simplicity, the circular shape of the cylinder is a challenge for numerical simulation.

The behavior of the flow can be laminar, transitional, or turbulent according to the Reynolds number value. In reference [5] a review is presented, which is referred that the key achievements in CFD simulation at Re=3900 were made in the development of Large Eddy Simulation and Detached Eddy Simulation. Direct Numerical Simulations (DNS) were carried out, in particular in references [6–8]. Numerical simulation in the current paper was also performed with DNS approach. Unsteady 3D flow over the cylinder is studied numerically using Computational Fluid Dynamics (CFD) approach based on the Navier-Stokes equations at Reynolds number Re ≈ 3900 which is the one most frequently encountered in literature.

Two flow types are considered, according to boundary conditions formulation: a flow around an isolated cylinder and a flow around a parallel cylinders cascade. It should be noted that the experimental data obtained in the study of the flow around the parallel cylinders cascade (see for example [9]) are used, in particular, to study correlation functions for isotropic turbulent flows. Numerical simulations have been carried out using the ANSYS Fluent code. Spectral properties of flow were analyzed using MATLAB.

2. Numerical technique
Incompressible unsteady 3D flow is considered in the configuration shown in Fig. 1. The streamwise and normal coordinates are denoted corresponding as x and y, and z is the spanwise coordinate. At the inflow boundary, a uniform flow field with the velocity vector \((u, v, w)^T = (1,0,0)^TV_\infty\) was prescribed, where \(V_\infty\) is the free stream velocity.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd
Figure 1. Computational domain (left) and O-mesh in the vicinity of the cylinder (right).

For performed DNS, the employed boundary conditions and computational mesh parameters are presented in Table 1. Referred to the cylinder wall parameters \( n_\theta \times n_r \times n_z \) and \( \Delta \theta^+ \times \Delta r^+ \times \Delta z^+ \) are the O-Grid cells number and wall-nearest grid cells (in wall-units), \( l_z \) is the spanwise size of the domain, and \( D \) is the diameter of the cylinder. Here \( \theta \) is the coordinate in the circumferential direction, and \( r \) is the coordinate in the radial direction. At the lateral sides of the computational domain (orthogonal to \( z \) coordinate), the “symmetry” boundary conditions were applied. Computational hexa-grid has been constructed in the commercial ANSYS ICEM grid generator.

### Table 1. Overview of the grid parameters based on the cylinder diameter \( D \).

| Case | Nb | Boundary type | \( n_\theta \times n_r \times n_z \) | \( \Delta \theta^+ \times \Delta r^+ \times \Delta z^+ \) | Grid total | \( l_z \) |
|------|----|---------------|-----------------|-----------------|-------------|------|
| A    | 10 | (1,0,0)\( V_\infty \) | 480 \times 150 \times 40 | 0.0131 \times 0.00137 \times 0.0785 | 16 \times 10^6 | \pi D |
| B    | 5  | Periodic      | 480 \times 150 \times 40 | 0.0131 \times 0.00137 \times 0.0785 | 16 \times 10^6 | \pi D |

In Fig. 1 (right) details of an O-Grid section employed in simulations are shown in the vicinity of the cylinder. The grid near the cylinder is stretched in the radial direction. The spanwise size along the \( z \) axis was chosen similarly to [3]. The distance 5\( D \) between the faces defining the periodic boundary conditions approximately corresponds to the distance between cylindrical rods in experimental investigations (see, for example, [9]).

The Navier–Stokes equations are solved using the ANSYS Fluent code with a second-order in space and a first order time discretization. The time step employed in the simulations is \( \Delta t = 0.0565 \frac{D}{V_\infty} \). Statistics were accumulated over approximately 100 vortex shedding cycles, \( T \approx 500 \frac{D}{V_\infty} \).

### 3. Results

Some of the important flow parameters from computations, and the available experimental data are summarized in Table 2. The mean drag coefficient \( C_D \) for Case A (isolated cylinder), base suction coefficient \( -C_{pb} \), separation angle \( \theta_{sep} \), and the cross flow Strouhal number \( St \) are found to be in satisfied agreement with the experimental data [10], [11].

### Table 2. Summary of cylinder flow computations compared to experimental data.

| Data from | \( C_D \) | \( -C_{pb} \) | \( St \) | \( \theta_{sep} \) |
|-----------|----------|-------------|--------|----------|
| Case A    | 0.96     | 0.91        | 0.224  | 85°      |
| Case B    | 1.125    | 1.06        | 0.255  | 84.2°    |
| Exp       | 0.99±0.05| 0.88±0.05   | 0.215±0.05 | 86±2°    |
Instantaneous velocity fields in sections across the cylinder axis are presented in Figure 2 for Cases A and B. Note the presence of both large and small structures in the wake, whereas the small size vortices are better visualized for Case A. From this figure, one can note also that the separation zone area for Case A is more in comparison to Case B.

![Figure 2](image_url)

**Figure 2.** Instantaneous velocity fields in section \((x, y, z = 0)\) across OZ axis for Cases A (left) and B (right).

Instantaneous recirculation zone sizes for Cases A and B can be estimated from distribution of the streamwise velocity in the \((x, y = 0, z)\) plane, Figure 3. By definition, the recirculation length \(L_r\) corresponds to the distance between the cylinder axis and the sign change of the centerline mean longitudinal velocity. From Figure 3 one can expect that the averaged recirculation zone length \(L_r \approx 1.6D\) approximately corresponds to reference data (see, for example [4]) for Cases A and B. Thus, for Case A the instantaneous recirculation zone length slightly exceeds the recirculation zone for Case B.

![Figure 3](image_url)

**Figure 3.** Instantaneous streamwise velocity in the \((x, y = 0, z)\) plane for Cases A (left) and B (right). There are contours from -1.5 m/s to 0. The black solid line shows the contour with \(u = 0\).

The vertical dashed line corresponds to the expected averaged boundary of the recirculation zone \(L_r \approx 1.6D\) [4]

Isosurfaces of streamwise vorticity are shown in Figure 4 at \(\omega = 5.31V_\infty/D\). The shear layers are formed, as the boundary layer on the cylinder surface detaches. Figure 4 shows quasiperiodic streamwise vortical structures in the near-wake region of the cylinder. These structures are similar (excluding vortical structures resolution) in shape and size to those described in references.
Visual analysis shows that the vortical structures for Case A reveal better spatial resolution compared to Case B. A similar conclusion can be made from a comparison of the instantaneous vorticity magnitude presented in Figure 5. The vortical structures tend to increase in size with increasing streamwise distance.

In Figures 4 and 5 the instantaneous streamwise vorticity highlights the two shear layers on the top and low sides of the cylinder. At the end of the recirculation zone, the shear layers become unstable and form primary vortices. At the distance $x = L_s$ downstream the flow the free-shear layers roll-up and a spanwise modulation in the shear layers can be observed. For Cases A and B, the shear layers are almost straight and laminar up to $L_s = 2D$ similarly to the instantaneous CFD flow visualizations, presented in [4].

For Case B the shear layers length tends to be less elongated. Ultimately both shear layers form the Karman vortex street (Figure 6) which begins further downstream with two shedding cycles visible in the domain area. Thus the vortex street shape is better resolved for Case A compared to Case B.

Figures 7, 8 show one-dimensional frequency spectra and Power Spectral Density for the velocity $v$ fluctuations at the distance $x = 4.5D$ on the centerline of the wake downstream from the cylinder. Power spectra are estimated with MATLAB.
Figure 7. FFT spectrum analysis of the normal velocity $v$ fluctuations for Case A (left) and Case B (right).

From Figure 7 it follows that the Karman vortex street frequency $F_{st}$ for Case B is slightly higher compared to Case A. Correspondingly the Strouhal numbers for Case B 14% higher compared to Case A. For Case A the St value corresponds to reference [11] and slightly over predicted compared to references [3], [4]. The power spectra are also able to reproduce the inertial subrange observed in the experimental researches.

Figure 8. Normal velocity $v$ Power Spectral Density for Case A (left), Case B (right).

In Figure 8 the signal frequency $F$ was made nondimensional using the Karman vortex street frequency $F_{st}$. Peaks at the fundamental $F/F_{st} = 1$ and second clearly visible harmonic $F/F_{st} = 3$ frequencies can be noted for the $v$-spectra. In general, the power spectra are consistent with the instantaneous flow visualizations, where the presence of small scales are better observed for Case A.

Conclusion

Unsteady incompressible 3D flow over a circular cylinder at Reynolds number of 3900 is studied numerically based on the Navier-Stokes equations using the ANSYS Fluent code. Two formulations of the problem were considered: with boundary conditions corresponding to the flow around an isolated cylinder and with periodic boundary conditions reproducing the flow behind a parallel circular cylinders grid. The O-Grid cells number and wall-nearest grid cells are similar for both formulations of the considered task. The simulation results reveal basic features of the flow over the cylinder: the presence of two free-shear layers and the vortex wake behind the cylinder for both boundary conditions types. The mean drag coefficient for the isolated cylinder, base suction coefficient, separation angle, and the cross flow Strouhal number are found to be in satisfying agreement with the experimental data. The vortical structures for the isolated cylinder reveal better spatial resolution compared to the periodical flow. One-dimensional frequency spectra and Power Spectral Density for the transversal velocity fluctuations at the distance on the centerline of the wake downstream from the cylinder. It was found that the Karman vortex street frequency for the periodical boundary conditions is slightly higher compared to the isolated cylinder.
Acknowledgments
This work was supported by the Russian Foundation for Basic Research (grant no. № 20-19-00548).

References
[1] Berger E and Wille R 1972 Annu. Rev. Fluid Mech. 4 313
[2] Williamson C H K 1996 Annu. Rev. Fluid Mech. 28 477
[3] Kravchenko A G and Moin P 2000 Physics of Fluids 12 2
[4] Parnaudeau P, Carlier J, Heitz D and Lamballais E 2008 Physics of Fluids 20 085101
[5] Molochnikov V M, Mikheev N I, Mikheev A N, Paereljy A A, Dushin N S and Dushina O A 2019 Fluid Dynamics Research 51 5
[6] Ma X, Karamonos G-S and Karniadakis G E 2000 J. Fluid Mech. 410
[7] Dong S, Karniadakis G E, Ekmekci A and Rockwell D 2006 J. Fluid Mech 569
[8] Wissink J G and Rodi W 2008 International Journal of Heat and Fluid Flow 29
[9] Stewart R W and Phil A A Trans. R. Soc. Lond. A 243
[10] Norberg C 1987 Publication 87/2 (Department of Applied Thermodynamics and Fluid Mechanics, Chalmers University of Technology, Sweden)
[11] Ong L and Wallace J 1996 Exp. Fluids 20 441