Journal of Wind Engineering & Industrial Aerodynamics 174 (2018) 39–49

Contents lists available at ScienceDirect

Journal of Wind Engineering & Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia

Direct numerical simulation of the flow around a rectangular cylinder at a moderately high Reynolds number

Andrea Cimarelli *, Adriano Leonforte, Diego Angeli

DISMI, University of Modena and Reggio Emilia, 42122, Reggio Emilia, Italy

A R T I C L E I N F O

Keywords:
Flow benchmark
Rectangular cylinder
Flow reattachment
Direct numerical simulation

A B S T R A C T

We report a Direct Numerical Simulation (DNS) of the flow around a rectangular cylinder with a chord-to-thickness ratio $B/D = 5$ and Reynolds number $Re = 3000$. Global and single-point statistics are analysed with particular attention to those relevant for industrial applications such as the behaviour of the mean pressure coefficient and of its variance. The mean and turbulent flow is also assessed. Three main recirculating regions are found and their dimensions and turbulence levels are characterized. The analysis extends also to the asymptotic recovery of the equilibrium conditions for self-similarity in the fully developed wake. Finally, by means of two-point statistics, the main unsteadinesses and the strong anisotropy of the flow are highlighted. The overall aim is to shed light on the main physical mechanisms driving the complex behaviour of separating and reattaching flows. Furthermore, we provide well-converged statistics not affected by turbulence modelling and mesh resolution issues. Hence, the present results can also be used to quantify the influence of numerical and modelling inaccuracies on relevant statistics for the applications.

1. Introduction

The flow around bluff bodies with sharp corners is known to be of overwhelming interest for several wind engineering applications (Tamura et al., 1998). The case of a rectangular cylinder encompasses the range of bluff bodies from a flat plate normal to the flow, to a square cylinder, and, finally, to a flat plate parallel to the flow, as its chord-to-thickness ratio is varied from zero to infinity. For these reasons, these kind of flows have been the subject of several numerical and experimental studies. Of particular interest for civil engineering applications is the case of slender bodies typical of buildings and structures. A peculiarity of these shapes resides in the fact that the flow exhibits a large-scale separation at the leading-edge and also a reattachment before the definitive separation at the trailing-edge. Indeed, while the shedding instability in the wake is observed in all bluff bodies, only long bluff bodies present further instabilities which are due to the separating and reattaching leading-edge shear layer. This leads to the formation of an additional shedding of large-scale vortices before the trailing edge. A detailed investigation into the nature of this separating and reattaching flow is found in Cherry et al. (1984). Despite the fact that these kind of flows have been the subject of several numerical and experimental studies, the topic is still attractive, as highlighted in a recent work by Bruno et al. (2014). From an applicative point of view, the interest is given by the fact that both experimental and numerical techniques appear to be unable to tackle the problem in an unequivocal way. Indeed, a large variability of results is found in the literature, even for global or first order statistics, see again the review of Bruno et al. (2014). The reason of these discrepancies is the high sensitivity of the flow on the test boundary conditions and measurement accuracy in experiments and on the turbulence model, numerical schemes and mesh properties in CFD analysis. Here, we focus on the numerical approach.

For low Reynolds numbers, $10^2 < Re < 10^3$ where $Re = U_\infty D/\nu$, U_∞ the free-stream velocity, D the rectangular cylinder thickness and ν the kinematic viscosity, the flow around rectangular cylinders has been studied in several works, see e.g. Nakamura et al. (1996), Ohya et al. (1992), Hourigan et al. (2001), and Tan et al. (2004). The main aim of the above mentioned works is the assessment of the main instabilities of the flow and of the self-sustaining mechanisms which generate them. Concerning the high Reynolds number regime, $Re > 10^4$, it is worth mentioning the works of Shimada and Ishihara (2002) and of Yu and Kareem (1998) where RANS (Reynolds Average Navier-Stokes) and LES (Large Eddy Simulation) techniques are respectively used. In this context, it is important to point out a benchmark activity on the aerodynamics of rectangular cylinders at Reynolds numbers of the order of

* Corresponding author.
E-mail address: andrea.cimarelli@unimore.it (A. Cimarelli).

https://doi.org/10.1016/j.jweia.2017.12.020
Received 12 June 2017; Received in revised form 11 December 2017; Accepted 15 December 2017

10^4, i.e. the BARC project (Benchmark on the Aerodynamics of a Rectangular 5:1 Cylinder) (Bartoli et al., 2011). Within this framework, a series of experiments and simulations have been conducted aiming at establishing reliable standards for the simulation and measurement of such a flow configuration, see e.g. Bruno et al. (2010), Mannini et al. (2011), Ricci et al. (2017) and Patruno et al. (2016) for its extension to non-null angles of attack.

As summarized in Bruno et al. (2014), the recent results within the BARC project are still characterized by a large scatter, thus highlighting that a clear picture of the combined influence of mesh resolution, turbulence model and boundary conditions on the flow statistics is still missing. One of the possible reasons is that, up to now, no reference data are available in the literature, i.e. experimental data obtained under well-defined boundary conditions (e.g. free-stream turbulence level) and unaffected by measurement errors, or numerical data not influenced by modelling and mesh resolution issues. Indeed, to the best of the authors' knowledge, no Direct Numerical Simulation (DNS) for sufficiently high Reynolds number has been performed in such a flow configuration. We found only two attempts in the literature. In the first one, Tamura et al. (1993) approached the problem by means of a finite difference technique at high Reynolds number, $Re = 10^4$. However, the grid resolution adopted was not fine enough to capture the smallest scales of motion and, hence, the simulation reported appears to be more an implicit LES than a DNS. More recently Hourigan et al. (2001) proposed a more accurate analysis through a spectral-element method. However, the DNS data reported refer to very low Reynolds numbers, namely from $Re = 350$ to $Re = 500$, and a fully developed turbulent state is not achieved.

In the present work we produce, for the first time, high-fidelity data of the flow around a rectangular cylinder with chord-to-thickness ratio $B/D = 5$ and Reynolds number $Re = 3000$. The study is aimed at understanding the main physical mechanisms driving the flow and at providing statistics, not affected by numerical issues, to be used for the validation and calibration of CFD techniques. For obvious computational cost reasons, the Reynolds number considered, $Re = 3{\cdot}10^3$, is smaller than the ones considered in the recent literature. However, let us point out that as shown by Sasaki and Kiya (1991), the flow develops the main turbulent structures typical of larger Reynolds numbers already for $Re > 380$. By further increasing Re, it is also found that the bubble length does not increase significantly anymore. It is also worth mentioning that, based on spectral arguments, Nakamura et al. (1991) argue that an asymptotic large Reynolds number regime is attained for $Re = 3000$ since for $Re > 3000$ the Strouhal number of the spectral peak does not increase anymore. Based on these results, we argue that the considered Reynolds number is sufficiently large to capture the main physical features observed at larger Reynolds numbers. As an example, the two main unsteadinesses observed by Kiya and Sasaki (1983, 1985) for very large Reynolds numbers and consisting of a shedding of vortices from the separation bubble and of a large scale oscillation encompassing the entire flow field, are found to be reproduced both qualitatively and quantitatively at the present Reynolds number (see section §3 for the details).

The paper is organized as follows. A description of the numerical simulation and of the statistical procedure is reported in section §2. The main statistical properties of the flow, with particular attention to those mostly debated in the BARC project, are shown in section §3. In order to rigorously assess the physical features characterizing the flow, single-point and two-point statistics are analysed in detail in sections §4 and §5. The paper is then closed by final remarks in section §6.

2. Direct Numerical Simulation and statistical convergence

A Direct Numerical Simulation has been performed to study the flow around a rectangular cylinder. The evolution of the flow is governed by the continuity and momentum equations,

$$\frac{\partial u_i}{\partial x_i} = 0$$
$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re}\frac{\partial^2 u_i}{\partial x_j x_j} \tag{1}$$

where $x = x_1$ $(u = u_1)$, $y = x_2$ $(v = u_2)$, $z = x_3$ $(w = u_3)$ are the streamwise, vertical and spanwise directions (velocity components), p is the pressure field and $Re = U_\infty D/\nu$ is the Reynolds number where U_∞ is the free-stream velocity, D is the thickness of the rectangular cylinder and ν is the kinematic viscosity. In accordance with the above equations, all the variables presented in the following will be reported in a dimensionless form by using D as length scale and D/U_∞ as time scale. A cell-centered finite volume method has been chosen to discretize the equations by means of the OpenFOAM® open source code (Weller et al., 1998). Time integration is performed by means of a second-order backward Euler implicit scheme while convective and diffusive fluxes at the volume faces are evaluated through a second-order central difference scheme. Finally, a pressure-implicit split-operator algorithm (Issa, 1986) is used to numerically solve the pressure-velocity coupling. Given the simple geometry of the problem, a block-structured Cartesian grid is adopted. Inlet-outlet boundary conditions are imposed in the streamwise direction. The inlet condition is a simple unperturbed flat velocity profile. The outlet boundary condition combines a Neumann/Dirichlet condition. In particular, a stress-free (zero gradient) condition is enforced when the flow exits the boundary, while a zero velocity vector is imposed when an inward flow is detected. The same kind of boundary condition is imposed in the vertical direction, the only difference being that in case of inward flow the imposed Dirichlet condition equals the free-stream inlet velocity. Finally, periodic conditions are imposed in the spanwise direction.

The flow case consists of a rectangular cylinder whose dimensions are $(L_x, L_y) = (5D, D)$. The Reynolds number considered is $Re = 3000$. The extent of the numerical domain is $(\mathscr{D}_x, \mathscr{D}_y, \mathscr{D}_z) = (112D, 50D, 5D)$ and is found large enough to not interfere with the flow dynamics, see the analysis of the spanwise correlation function shown in section §5. A sketch of the system configuration and of the reference coordinate system is reported in Fig. 1. The structured Cartesian grid employed is composed by $1.5{\cdot}10^7$ volumes. A multi block-structured approach is used by employing 5 main blocks characterized by a stepwise variation of the number of volumes in the spanwise direction. In particular, in the inner block, the number of volumes in the (x, z)-plane above the rectangle is $(Nx, Nz) = (128, 144)$ in the streamwise and spanwise direction, respectively. The volume distribution is homogeneous in the spanwise direction while in the streamwise and vertical directions a geometric progression is adopted, $\Delta x_i = k_x^{i-1}\Delta x_1$ and $\Delta y_j = k_y^{j-1}\Delta y_1$ with $k_x = 1.06$, $k_y = 1.04$, $\Delta x_1 = 0.004$ and $\Delta y_1 = 0.004$. This approach is used to obtain higher resolution levels in the near-wall leading- and trailing-edge regions. Such a practice leads to a mean wall resolution of $(\overline{\Delta x^+}, \overline{\Delta y^+}, \overline{\Delta z^+}) = (6.1, 0.31, 5.41)$, where $\overline{(\cdot)}$ denotes the streamwise average along the rectangle length and the superscript $+$ implies

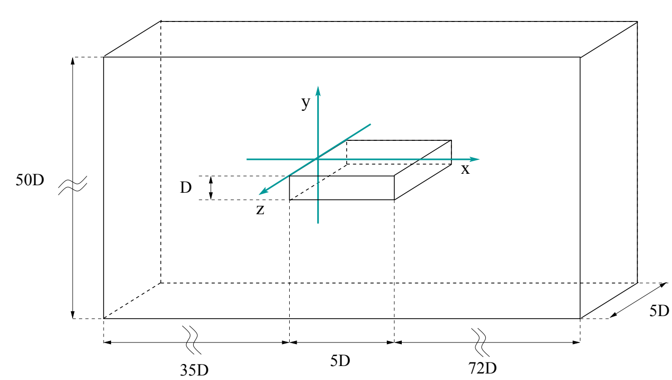

Fig. 1. Configuration of the system.