J. Wind Eng. Ind. Aerodyn. 151 (2016) 146–157

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

Numerical simulation of a 5:1 rectangular cylinder at non-null angles of attack

L. Patruno, M. Ricci*, S. de Miranda, F. Ubertini

DICAM, University of Bologna, Viale Risorgimento 2, 40136 Bologna, Italy

A R T I C L E I N F O

Article history:
Received 19 March 2015
Received in revised form
15 January 2016
Accepted 17 January 2016
Available online 1 March 2016

Keywords:
LES
Bluff body
Rectangular cylinder

ABSTRACT

In this paper, the flow around a rectangular cylinder with aspect ratio 5:1 is studied at three attack angles by means of both LES and URANS simulations. Despite its geometric simplicity, such a rectangular shape is characterized by many aerodynamic phenomena typical of reattached flows around bluff bodies and has been recently widely studied as a prototype of flows encountered around shapes of technical interest like, for example, bridge deck sections. The present study is aimed at providing insight in the modifications occurring in the flow at small incidence angles, by comparing the accuracy of the two aforementioned simulation strategies in reproducing them. Some recurring biases observed in URANS simulations are illustrated and their origin discussed. Results are presented in terms of flow bulk parameters and pressure distributions and systematic comparison with available experimental data is provided. Finally, the effects of small incidence angles on along span flow correlations are investigated and Covariance Proper Transformation is used in order to further characterize the flow dynamic behavior.

1. Introduction

The incompressible high Reynolds number, *Re*, flow around the rectangular prism with aspect ratio 5:1 at null attack angle has been extensively studied in recent literature, aiming at constituting a standard benchmark in the wind engineering community for the comparison between experimental data and Computational Fluid Dynamics (CFD) simulations. In particular, the BARC project (Bartoli et al., 2008) provided a convenient platform able to concentrate the efforts of numerous research groups toward the common objective of investigating both the predictive capability of computational models and the variability of the results obtained with such techniques. In fact, despite its geometric simplicity, the flow around the 5:1 rectangular cylinder is characterized by strong detachments at the leading edges and reattachments which induce the development of unsteady separation bubbles on the along wind sides. As a consequence, from the aerodynamic point of view, such flow is well representative of configurations often found around more complex geometries like, for example, bridge decks and high-rise buildings. The length and dynamic behavior of such flow features are strongly related to the stability conditions of the detached shear layers which, in turn, are deeply affected by the adopted simulation strategy.

Without a doubt, one of the most important merits of BARC has been to investigate the variability of the results obtained by different authors and simulation strategies and analyze them on a statistical base. Not surprisingly, due to the aforementioned flow features, the overview of the first four years of activity of the BARC project highlighted a remarkable scattering of the results obtained with CFD simulations even for what concerns first order statistics of the flow field (Bruno et al., 2014). In particular, it appears that the simulation strategy can deeply affect the reattachment length so leading to a great variability of the predicted flow topology: the relative importance of the turbulence model, numerical schemes and mesh size is still difficult to be evaluated at the present stage.

Despite the high dispersion of the data, by analyzing the statistics of the pressure distributions obtained with LES and URANS simulations, it is possible to observe that not only do the two lead to comparable accuracy in terms of time averaged pressure distributions, but they are often comparable also in terms of the standard deviation of the pressure fluctuations.

Notwithstanding the described flow complexity, it is noticed that the null attack angle, as prescribed by the BARC main setup (Bartoli et al., 2008), represents a very special case due to the presence of symmetry which, at least statistically, represents a constraint to the flow development (Carassale, 2009). The breaking of symmetry in nominally symmetric conditions has been extensively documented within the BARC project and can be still considered an open issue (Bruno et al., 2012).

In this paper, the flow around a 5:1 fixed rectangular prism is studied at non-null attack angles by means of both LES and URANS

---

* Corresponding author.
E-mail address: mattia.ricci10@unibo.it (M. Ricci).

http://dx.doi.org/10.1016/j.jweia.2016.01.008
0167-6105/

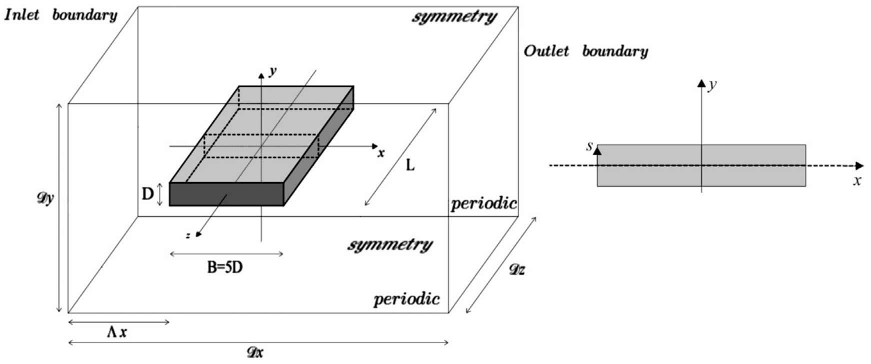

Fig. 1. Model and domain geometry adopted for computational studies (Bartoli et al., 2008).

simulations aiming at providing insight in the flow modifications at small incidence and testing the capabilities of CFD techniques in reproducing them. The paper is intended to be, on one side, a self-contained work which investigates and compares the performances of LES and URANS simulations with special attention on the biases observed with respect to experimental results and, on the other side, a first explorative contribution to the study of the 5:1 rectangular cylinder at non-null attack angle, as required by the sensitivity studies proposed within BARC.

The paper is organized as follows: Section 2 briefly describes the setup used to extract experimental data while Section 3 describes the adopted numerical model for both LES and URANS simulations. Results obtained at three attack angles (namely 0°, 1° and 4°) with the described numerical settings are presented and discussed in Section 4. In particular, central section and span-averaged pressure coefficients statistics are reported. Then, along-span correlations and Covariance Proper Transformation are used to further characterize and provide insight in the flow dynamics. Finally, in Section 5 some conclusions are drawn.

2. Experimental setup

In this study, the obtained numerical results are compared mainly to the experimental data (Personal communication from CRIACIV, 2015) obtained in the open-circuit boundary layer wind tunnel of CRIACIV laboratory located in Prato (Italy).

The experimental investigations employed an aluminium model with chord, B, equal to 300 mm, height, D, equal to 60 mm and span, L, equal to 2380 mm, horizontally mounted and characterized by sharp edges and smooth surfaces (Bruno et al., 2014). The model was equipped with 62 pressure taps and two 32-channels PSI miniaturized piezoelectric scanners recorded simultaneous pressures at a sampling rate of 500 Hz (Mannini et al., 2015).

In the experimental tests, three nose-up angles of attack were considered, namely 0°, 1° and 4°. The blockage ratio was equal to 3.75%, 4.1% and 5.05% in the three analyzed configurations, respectively. The alignment of the model in the wind tunnel was obtained by rotating it of 0.4° with respect to the horizontal plane in order to reach a symmetric pressure distribution on the top and bottom faces. Smooth inlet was considered, with a turbulence intensity equal to 0.6%. The Re number based on D was equal to 5.5×10^4 for 0°, while it was equal to 6.28×10^4 for 1° and 4°.

For what concerns the flow bulk parameters, in addition to data obtained at the CRIACIV laboratory, a comparison with experimental results obtained by Schewe (2013) is provided. In this case, results have been obtained in the pressurized wind tunnel of the Institute for Aeroelasticity located in Göttingen.

3. Computational model

In this section, the numerical settings adopted for the two investigated simulation strategies are reported. According to the BARC main setup, the Re number based on D is set to 2.7×10^4 and smooth inlet is adopted.

It must be noticed that numerous evidences of the flow sensitivity to the Re number, even around sharp edged bluff bodies, can be found in the literature (Schewe and Larse, 1998; Schewe, 2001, 2009). In particular, Schewe (2013) studied the Reynolds number effects around a rectangular cylinder with aspect ratio 5:1 showing that, in this case, only a slight dependence of the aerodynamic coefficients is found in the range from $Re=2.7\times10^4$ to $Re=5.5\times10^4$ for attack angles lower than 6°. Therefore, although appreciable dependencies can be expected outside the aforementioned Re number range, the obtained results have been assumed compatible with the experimental data used for comparison.

According to the nomenclature introduced in Fig. 1, the computational domain is such that $D_x=40B$ and $D_y=30B$ leading to a blockage ratio equal to 0.67% at null attack angle and, thus, negligible blockage effects. Additionally, the distance of the front face of the prism from the inlet, Λ_x, is set equal to 16B in order to avoid the influence of boundary conditions on the flow development around the immersed body. Pressure–velocity coupling is obtained for all analyses by using the well-known PIMPLE algorithm (Ferzinger and Peric , 1999; Versteeg and Malalasekera , 1995; OpenFOAM, 2015) while the time advancement is the implicit two-step second order Backward Differentiation Formulae (BDF) in accordance with Bruno et al. (2012).

The prism has been equipped with 2500 pressure monitors and data are acquired at each time step. All simulations have been performed by using the open source Finite Volume software OpenFOAM on 120 CPUs at the CINECA PLX cluster (274 nodes, 2-six cores Intel Westmere 2.40 GHz processors with 48 GB RAM per node).

3.1. Numerical setup for LES

The computational domain size and grid resolution have been defined according to the guidelines provided by Bruno et al. (2014). Close to the solid boundaries, a structured mesh, characterized by along wind dimension $\delta_x/B=2.5\times10^{-3}$ and cross wind dimension $\delta_y/B=1.5\times10^{-3}$, is adopted while the first cell height, n_w, is equal to $5.0\times10^{-4}B$, leading to a maximum non-dimensional wall distance y^+ equal to 5.7. Such value is recorded in the proximity of the leading edge, while its mean value equals 1.6. The mesh stretching ratio close to the prism in the wall normal direction is set equal to 1.1 (see Fig. 2(a)). Outside the boundary layer, the mesh is unstructured quad dominated and its size is slowly coarsened up to approximately $\delta_x/B=\delta_y/B=1.8\times10^{-2}$.