The 8th International Conference on Jets, Wakes and Separated Flows, ICJWSF-2024
September 23-25, 2024, Firenze, Italy

Flow over a 5:1 rectangular cylinder at moderate Reynolds numbers: Comparison between DNS, LES, and experiments

M. Morello[1*], G. Lunghi[1], A. Mariotti[1], M.V. Salvetti[1], R. Corsini[2], A. Cimarelli[2], E. Stalio[2]

1: Dept. of Civil and Industrial Engineering, University of Pisa, Pisa, Italy
2: Dept. of Engineering ”Enzo Ferrari”, University of Modena and Reggio Emilia, Modena, Italy

[*] Corresponding author: mario.morello@phd.unipi.it

Abstract

This study investigates the flow dynamics around a 5:1 rectangular cylinder at moderate Reynolds numbers, focusing on a comprehensive comparison between Direct Numerical Simulations (DNS) and experimental results at a Reynolds number of $Re = 14000$. This represents the first direct comparison at this Reynolds number, offering crucial cross-validation and revealing the influence of different setup conditions, such as the presence of freestream turbulence in the experiments. Both experimental and DNS data are further employed to assess the accuracy of Large-Eddy Simulations (LES). Additionally, a sensitivity analysis using LES explores the effects of upstream edge rounding and computational domain size on the flow characteristics. The findings demonstrate excellent agreement between DNS and experimental data regarding the length and shape of the mean recirculation region along the lateral side of the cylinder. However, DNS predicts higher pressure fluctuations near the reattachment point of the mean flow on the cylinder's side. For the LES results, the following conclusions can be drawn: (i) at moderate Reynolds numbers, LES effectively captures the mean flow topology on the cylinder's side; (ii) the treatment of perfectly sharp edges is less critical at lower Reynolds numbers; (iii) LES does not accurately capture the transition and dynamics of the detaching shear layers; and (iv) the observed discrepancies in pressure fluctuation distribution between LES and DNS cannot be solely attributed to differences in computational domain size.

Keyword:*Large-Eddy Simulations, elongated rectangular cylinders, upstream edge rounding, computational domain size, moderate Reynolds number*

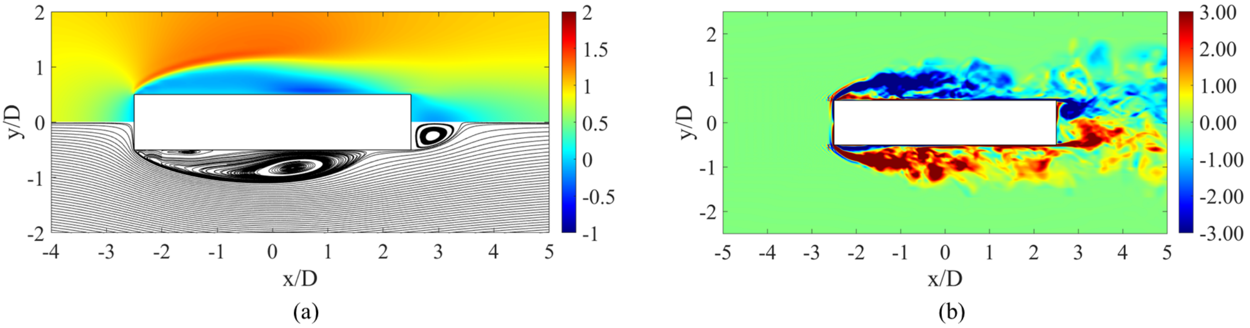


Fig. 1: Flow behavior around the 5:1 rectangular cylinder at $Re = 14000$: (a) time-average of the streamwise-velocity component (upper panel) and mean-flow streamlines (bottom panel); (b) instantaneous spanwise-average of the vorticity in the spanwise direction.

1. Introduction

Elongated rectangular cylinders represent simplified geometries that are critical for studying real civil structures, such as tall buildings and bridge decks. These structures experience complex aerodynamic interactions, which can be better understood by analyzing the flow around such simplified models. The high-Reynolds number flow around a 5:1 rectangular cylinder at zero angle of attack is the central focus of the international Benchmark on the Aerodynamics of a Rectangular 5:1 Cylinder (BARC). This benchmark serves as a crucial reference for both experimental and numerical studies aimed at elucidating the aerodynamic behavior of these structures.

The flow around the cylinder is characterized by shear-layer separation at the upstream edges (see, e.g., Fig. 1). This separation leads to the forming of vortical structures due to Kelvin-Helmholtz instability, a phenomenon occurring when there is velocity shear within a continuous fluid or between two different fluids. These vortices are then convected downstream, where they interact with other vortical structures. This interaction results in the formation of the classical Von Karman vortex street in the near wake, a repeating pattern of swirling vortices caused by the unsteady separation of flow over bluff bodies. The mean flow topology reveals a closed separated region along the cylinder's lateral side, indicating complex flow behavior that requires thorough investigation.

Numerous numerical simulations, including LES and hybrid approaches, as well as various experimental studies conducted in different facilities (e.g., [6] and [8]), have been carried out within the BARC benchmark. These studies have been performed at Reynolds numbers ranging from 20000 to 110000, based on the cylinder depth D and the freestream velocity. Contributions to the BARC benchmark, as reviewed in [1], have demonstrated significant variability in the mean flow topology around the cylinder, leading to notable discrepancies in key quantities, such as the distribution of mean and fluctuating pressures on the cylinder's sides. Despite extensive research efforts, no definitive explanations for these discrepancies have been proposed. In contrast, DNSs have been conducted at lower Reynolds numbers. Specifically, DNS studies have been performed at $Re = 3000$ as reported by [3] and [2], and more recently at $Re = 8000$ and $Re = 14000$ as detailed in [4]. These numerical investigations provide a more detailed and precise understanding of the flow characteristics, serving as a crucial reference point for validating other simulation approaches.

The present study investigates the flow over a 5:1 rectangular cylinder at a moderate Reynolds number of $Re = 14000$. The primary aim is to compare and cross-validate experimental and LES findings with the DNS results of [4] at the same Reynolds number. Additionally, a systematic analysis is conducted to assess the impact of varying the LES setup, particularly with respect to upstream edge rounding and computational domain size, on the resulting flow patterns around the cylinder. By exploring these factors, the study aims to deepen the understanding of the aerodynamic behavior of rectangular cylinders and address discrepancies identified in previous research within the BARC benchmark.

2. Experimental set-up and methodology

The experimental campaign was conducted in a closed-return subsonic wind tunnel at the University of Pisa. This wind tunnel is equipped with a circular open test section, having a diameter of 1.1 meters and a length of 1.42 meters, while maintaining a freestream turbulence level of 0.9%. The model employed in these experiments is a 5:1 hollow rectangular cylinder made of aluminum alloy, with dimensions of 200 mm in the streamwise directionn (x), 40 mm in the crossflow direction (y), and 800 mm in the spanwise direction (z). To minimize three-dimensional effects, two end plates were positioned at the spanwise ends of the model, located at $z/D = \pm 10$.

The model is equipped with 495 pressure taps, with 72 of these located along the spanwise centerline ($z/D = 0$). The experiments are conducted at a Reynolds number of 14000, based on the freestream velocity and the cylinder depth. Differential pressures are recorded using two Pressure Systems ESP-16HD miniature electronic pressure scanners, which are housed directly inside the model. Velocity measurements are obtained using an IFA AN 1003 A.A. Lab System hot-wire anemometry module, equipped with Dantec 55P11 probes. These probes are capable of movement in all directions with an accuracy of 0.1 mm, ensuring precise and reliable measurements.

For each realization, the sampling frequency is set at 16000 Hz, with an acquisition duration of 32.768 seconds. The curvature radius of the model's upstream edges is measured using a digital microscope, specifically the RS PRO model. This detailed experimental setup facilitates accurate and comprehensive data

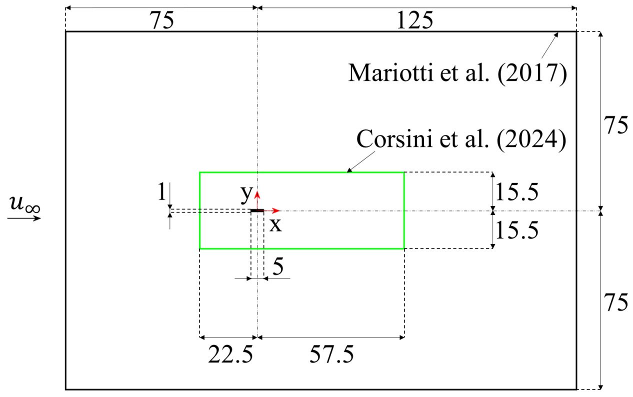


Fig. 2: Sketch of the cross-flow section of the computational domains in the DNS in [4] with the green line, and the LES in [7] with the black line.

collection, which is critical for a thorough understanding of the aerodynamic behavior of the rectangular cylinder under investigation.

3. Direct Numerical Simulation set-up

The numerical simulation of turbulent flow around the 5:1 rectangular cylinder was conducted using the open-source code Nek5000, which utilizes a high-order spectral element method. Within each element, the velocity components and pressure are expanded using the tensor product of Legendre polynomials of order N and $N-2$, respectively. For the present DNS, the polynomial order was set to $N=7$. Time integration was performed using a second-order backward differentiation scheme, combined with a second-order extrapolation scheme for handling the nonlinear terms. The non-dimensional time step was maintained at $\Delta t = 2.6\times10^{-4}$ throughout the simulation, ensuring that the Courant-Friedrichs-Lewy (CFL) condition remained below 0.5 at all points in the domain.

Regarding boundary conditions, a constant streamwise velocity profile was imposed at the inflow without turbulence, traction-free boundary conditions were applied at the outflow and in the cross-stream direction, and a no-slip condition was enforced on the cylinder surfaces. The spanwise boundaries were treated as periodic.

The Reynolds number for the simulation matched that of the experiments, set at 14,000. The computational domain dimensions were $80D\times31D\times5D$ in the streamwise, cross-stream, and spanwise directions, respectively. The upstream face of the cylinder was positioned $20D$ from the inflow and centered in the cross-stream direction. The simulation employed $N_e = 8988000$ elements, corresponding to approximately 3 billion degrees of freedom per time step per unknown. The element distribution was homogeneous in the spanwise direction but refined in the wall-normal direction as it approached the cylinder surfaces. The smallest grid spacing at the edges of the cylinder was ($\delta x_{\min}$; $\delta y_{\min}$; δz) = (0.0021; 0.0021; 0.007). These spacings were determined as the distance between $N+1$ evenly spaced nodes within the spectral element. In the near-wall region, the spatial resolution satisfied the following characteristics: (δx^+; δy_w^+; δz^+)$_{\max}$ = (4.1; 0.66; 5.1), where the superscript $+$ denotes normalization in viscous units and δy_w is the distance from the wall to the second computational node. In turbulent regions, the ratio between the grid spacings and the Kolmogorov scale η is at most ($\delta x/\eta$; $\delta y/\eta$; $\delta z/\eta$)$_{\max}$ = (4.2; 4.6; 6.3).

Statistics were collected over a period of $250D/U_\infty$ while the flow was statistically stationary, and then averaged in the spanwise direction and over the $x-z$ symmetry plane. This rigorous setup enabled the collection of highly detailed and accurate simulation data, facilitating a deeper understanding of the turbulent flow dynamics around the rectangular cylinder.

4. Large-Eddy Simulation set-up and methodology

LESs were conducted using the open-source code Nek5000, following similar procedures as those used in Direct Numerical Simulations (DNS). The LES configurations were based on setups described in [7] and [9]. Each spectral element is either rectangular or an appropriate coordinate mapping of that shape. Within these elements, the velocity and pressure were expanded using Legendre polynomials of order N and $N-2$

Tab. 1: Configurations examined.

Case number	**Size of the computational domain** $x/D \times y/D \times z/D$	***r/D***
Case 1	$200D \times 150D \times 5D$	0
Case 2	$200D \times 150D \times 5D$	0.0005
Case 3	$200D \times 150D \times 5D$	0.0037
Case 4	$80D \times 31D \times 5D$	0

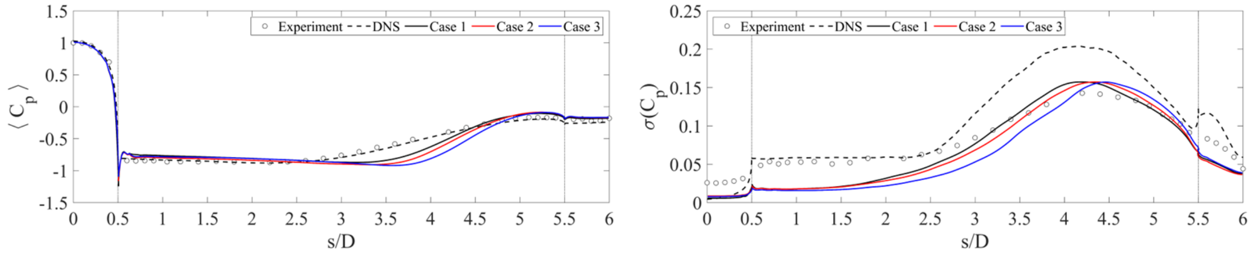
respectively; in this study, $N = 6$, consistent with [7]. A third-order backward finite-difference scheme, utilizing a high-order splitting method, was employed for time integration. The spectral element distribution in LES matched that applied in [9]. Near the cylinder, the element size was $\Delta x/D = \Delta y/D = 0.125$ in the streamwise and crossflow directions, while the spanwise direction maintained a uniform size of $\Delta z/D = 0.558$. Since the grid resolution was insufficient to capture all turbulent scales at the given Reynolds number, a low-pass filter was applied to the velocity field in the modal space, introducing dissipation only at the highest resolved modes. This served as a form of SubGrid-Scale (SGS) dissipation. At the inlet, a uniform, turbulence-free velocity profile (smooth flow) was imposed, while traction-free boundary conditions were applied at the outflow and the domain's upper and lower boundaries. A no-slip condition was enforced on the cylinder's surface, with periodic boundary conditions used in the spanwise direction.

This study aims to systematically analyze the effects of two key parameters: upstream edge rounding and computational domain size. Three different rounding configurations for the upstream edges were tested: $r/D = 0$, $r/D = 0.0005$, and $r/D = 0.0037$, where r is the radius. Additionally, two computational domain configurations were examined: one consistent with [4], and the other similar to that used in [9], with dimensions $-75 \leq x/D \leq 125$, $-75 \leq y/D \leq 75$, and $0 \leq z/D \leq 5$. In both cases, the cylinder's center was located at $x/D = 0$ and $y/D = 0$. The size difference between the two domains is illustrated in Fig. 2. Table 1 lists the examined configurations and assigns them identifying codes.

These setups allow for a thorough investigation of how different configurations impact the flow characteristics around the 5:1 rectangular cylinder at a Reynolds number of 14000.

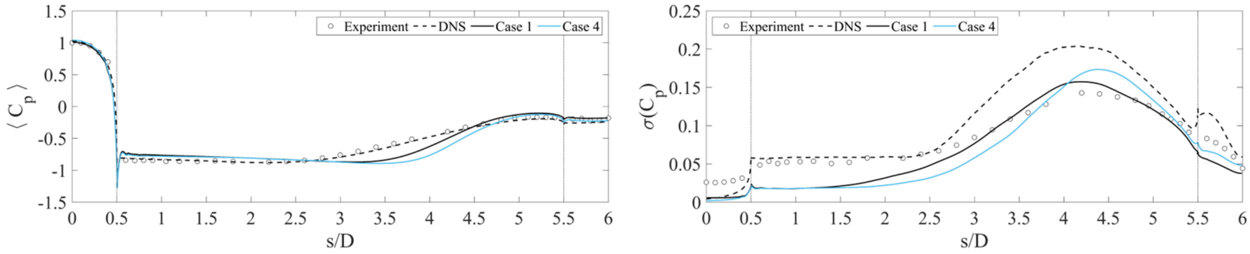
5. Results and Discussion

A preliminary comparison of experimental data, Direct Numerical Simulation (DNS), and Large-Eddy Simulation (LES) results is presented in Fig. 3. This figure shows the distribution of the time-averaged pressure coefficient, $\langle C_p \rangle$, and its standard deviation, $\sigma(C_p)$, along the lateral side of the rectangular cylinder with sharp upstream edges. Results for configurations with rounded upstream edges are also included. The DNS predictions align perfectly with the experimental results for both the mean pressure coefficient distribution and the peak position of the standard deviation. This agreement suggests that the experimental setup accurately predicts the streamwise extent of the mean recirculation region, as inferred from the relationship between the location of $(\sigma(C_p))_{\max}$ and the length of this recirculation region [5]. However, discrepancies



(a) (b)

Fig. 3: (a) Time-averaged pressure coefficient and (b) its standard deviation on the rectangular cylinder surface. Effect of rounding of the upstream edges.



(a) (b)

Fig. 4: (a) Time-averaged pressure coefficient and (b) its standard deviation on the rectangular cylinder surface. Effect of computational domain size.

in the maximum value of $\sigma(C_p)$ warrant further investigation.

The standard deviation of the pressure coefficient shows good agreement between LES and DNS regarding the peak position of the fluctuations. Nevertheless, differences are evident in the spatial evolution of the mean pressure coefficient, indicating variations in the shape of the mean recirculation region. Additionally, LES exhibits lower pressure fluctuations in the upstream portion of the cylinder side compared to DNS. Furthermore, Figure 3 illustrates that the influence of upstream edge rounding on the pressure field is less pronounced at moderate Reynolds numbers ($Re = 14000$) compared to the effects observed by [9] at $Re = 40000$.

Figure 4 illustrates the pressure distribution and its fluctuations on the model's surface, emphasizing their sensitivity to three parameters: computational domain size, spectral element size near the cylinder, and polynomial order. Examining the effect of computational domain size, we observe that a reduction in domain size results in a slight increase in the peak of the pressure fluctuation standard deviation, $\sigma(C_p)$. However, this increase is insufficient to account for the higher peak observed in DNS results.

Further results from the sensitivity analysis will be presented in the final discussion.

6. Conclusions

This study investigates the flow dynamics around a 5:1 rectangular cylinder at a moderate Reynolds number of $Re = 14000$. We compared results from various investigative methods, specifically Direct Numerical Simulation (DNS), Large-Eddy Simulation (LES), and experimental data. Additionally, a parametric study was conducted using LES to assess the effects of upstream edge rounding and computational domain size.

The comparison between DNS and experimental data reveals that: (i) there is generally good agreement between DNS and experimental results, despite variations in freestream turbulence; and (ii) the discrepancies in the standard deviation of the pressure coefficient, $\sigma(C_p)$, along the cylinder's side require further investigation.

Regarding the LES results, we find that: (i) at $Re = 14000$, LES accurately predicts the length of the mean recirculation region along the side of the cylinder; (ii) the impact of small rounding of the leading edges on the aerodynamic characteristics is less significant at $Re = 14000$ compared to observations at $Re = 40000$; (iii) LES struggles to capture the transition and dynamics of the detaching shear layers; and (iv) the difference in computational domain size between LES and DNS does not seem to explain the higher peak of $\sigma(C_p)$ observed in DNS.

Future studies could focus on evaluating the effect of grid resolution, both by varying the size of spectral elements (h-refinement) and the polynomial order N (p-refinement), calibrating simulation parameters, incorporating dissipation mechanisms in implicit LES, or exploring alternative SubGrid-Scale (SGS) models.

7. Acknowledgments

We acknowledge the CINECA award under the ISCRA initiative, for the availability of high-performance computing resources and support.

References

[1] L. Bruno, M. V. Salvetti, and F. Ricciardelli. “Benchmark on the aerodynamics of a rectangular 5: 1 cylinder: an overview after the first four years of activity”. In: *J. Wind Eng. Ind. Aerod.* 126 (2014), pp. 87–106.

[2] A. Chiarini and M. Quadrio. “The turbulent flow over the BARC rectangular cylinder: a DNS study”. In: *Flow Turbul. Combust.* 107 (2021), pp. 875–899.

[3] A. Cimarelli, A. Leonforte, and D. Angeli. “Direct numerical simulation of the flow around a rectangular cylinder at a moderately high Reynolds number”. In: *J. Wind Eng. Ind. Aerod.* 174 (2018), pp. 39–49.

[4] R. Corsini, A. Cimarelli, and E. Stalio. “DNS of the Flow About a 5: 1 Rectangular Body with Sharp Corners”. In: *Marchioli, C. et al. (eds) Direct and Large Eddy Simulation XIII. ERCOFTAC Series* 31 (2024), pp. 9–16.

[5] G. Lunghi et al. “Impact of the lateral mean recirculation characteristics on the near-wake and bulk quantities of the BARC configuration”. In: *Wind Struct.* 34(1) (2022), pp. 115–125.

[6] C. Mannini et al. “The effects of free-stream turbulence and angle of attack on the aerodynamics of a cylinder with rectangular 5: 1 cross section”. In: *J. Wind Eng. Ind. Aerod.* 161 (2017), pp. 42–58.

[7] A. Mariotti, L. Siconolfi, and M. V. Salvetti. “Stochastic sensitivity analysis of large-eddy simulation predictions of the flow around a 5: 1 rectangular cylinder”. In: *Eur. J. Mech./B Fluids* 62 (2017), pp. 149–165.

[8] E. Pasqualetto et al. “Experimental characterization of the lateral and near-wake flow for the BARC configuration”. In: *Wind Struct.* 34(1) (2022), pp. 101–113.

[9] B. Rocchio, A. Mariotti, and M. V. Salvetti. “Flow around a 5: 1 rectangular cylinder: Effects of upstream-edge rounding”. In: *J. Wind Eng. Ind. Aerod.* 204 (2020), p. 104237.