Analysis of the separated flow around a 5:1 rectangular cylinder through computational simulation

L. Bruno¹, N. Coste², D. Fransos²

¹Dipartimento di Ingegneria Strutturale e Geotecnica, Politecnico di Torino - luca.bruno@polito.it
- Viale Mattioli 39, 10126 Torino, Italy
²OptiFlow Company - coste@optiflow.fr, fransos@optiflow.fr
Bât. Azurburo, 27, boulevard Charles Moretti, 13014 Marseille, France

Keywords: Computational Wind Engineering, Large Eddy Simulation, rectangular cylinder, BARC benchmark

ABSTRACT

The aim of this paper is to provide a contribution to the analysis of the 3D, high Reynolds number, turbulent, separated and reattached flow around a fixed sharp-edged rectangular cylinder with a chord-to-depth ratio equal to 5. The present computational study is developed in the frame of the Benchmark on the Aerodynamics of a Rectangular Cylinder (BARC).
First, the adopted flow modelling and computational approach are described. Second, some of the 2D mechanisms that are responsible for the variation of the fluctuating pressure on the side surface are scrutinised: the computational approach post-processing facilities are employed to point out the flow structures which mainly affect the vortex formation and shedding past the separation point and to look for significant relationships between them and the pressure field on the cylinder side surface.

1. INTRODUCTION

The aerodynamic behaviour of rectangular cylinders has attracted the attention of the scientific
community since the experimental reference works of Okajima (1982) and Norberg (1993). On one hand, both the two dimensional (2D) and three dimensional (3D) features of the low-Reynolds number flow around rectangular cylinders has been clarified in several studies, e.g. in Nakamura et al. (1996), Hourigan et al. (2001), Tan et al. (2004). On the other hand, the high-Reynolds number flow (i.e. \( \text{Re} \geq 1.0 \times 10^4 \)) has been studied by means of both experimental and computational approaches, with emphasis on its dependence on the chord-to-depth ratio, e.g. in Yu & Kareem (1996,1998), Shimada & Ishihara (2002). In particular, the high Re number flow around the square cylinder has been widely investigated since the beginning of the Nineties, on the basis of some detailed experimental measurements (Durao et al. 1988, Lyn and Rodi 1994) and in the framework of a benchmark study promoted in the Computational Fluid Dynamics community (Ercotiac Test Case LES2, Rodi et al. 1997). The large amount of sensitivity studies performed and the results obtained for this case-study can provide a useful background knowledge for the study of the aerodynamic behaviour of more elongated sections, despite the involved fluid flow phenomena are quite different, mainly because of the boundary layer reattachment along the section side surface and the interaction between vortical structures shed from the leading and trailing edges.

Recently, a Benchmark on the Aerodynamics of a 5:1 Rectangular Cylinder (\texttt{BARC}) has been proposed in order to provide a contribution to the analysis of the high-Reynolds number, turbulent, separated flow around a fixed rectangular cylinder with chord-to-depth ratio equal to 5 (Bartoli et al. 2008). This ratio is far enough from those at which discontinuities in the aerodynamic regime arise, i.e. the 2.8 and 6 ratios (Shimada & Ishihara 2002), in order to avoid the introduction of further difficulties in the study. For this benchmark, the depth-based Reynolds number has to be in the range of \( 2.0 \times 10^4 \leq \text{Re}_D \leq 6.0 \times 10^4 \), the oncoming flow has to be set parallel to the base of the rectangle (\( \alpha = 0 \)) and the maximum intensity of the longitudinal component of turbulence has to be \( I_u = 0.01 \). The selected cylinder is considered as a representative benchmark of a bridge deck or high-rise building elongated section.

In the perspective of the benchmark, a computational exploratory study has been performed by the present authors (Bruno et al. 2008) in order to focus on three main aspects. First, the obtained main aerodynamic integral parameters are compared with other results proposed in literature (Yu & Kareem 1996, 1998, Shimada & Ishihara 2002). The overall simulated aerodynamic behaviour seems to well agree with the reference results, even though the data about the fluctuating force components are sometimes scattered: the expected parameter sensitivity to physical incoming flow conditions (e.g. Re number, turbulence intensity and integral length scale), experimental set-up conditions and/or computational model components (e.g. turbulence modelling, numerical approaches) could be systematically addressed in future researches in the frame of the \texttt{BARC} activity. Second, the 3D features of the flow around the nominally 2D bluff cylinder has been evaluated by means of two different techniques. The spanwise coherence of the pressure field on the cylinder lateral surface qualitatively agrees with the results in literature (Matsumoto et al. 2003), even though it is generally underestimated. The Proper Orthogonal Decomposition (POD), often adopted in literature for the analysis and synthesis of random wind pressure fields, especially on high-rise buildings (e.g. Holmes 1990, Tamura et al. 1999) has been also applied to the side-surface fluctuating pressure field. It has shown that, even though the 3D flow features are not negligible, the main phenomena which drive the aerodynamic forces remain 2D. Finally, a first attempt has been made in Bruno et al. (2008) to approach some remaining difficulties in describing the expected complex flow phenomena around the cylinder and in relating such phenomena to the fluctuating aerodynamic forces acting on the cylinder itself. The computational approach post-processing facilities has been employed to look for significant relationships between flow structures, pressure field and aerodynamic forces. As a result, a guess conceptual partition of the side-surface has been proposed and the so-called “mean pressure recovery” region has been identified as the one that gives the most significant contribution to the overall lift force.

The present study aims to give a deeper insight in the computational results previously obtained, with special emphasis to the relationships between the fluctuating pressure field along the side surface and the velocity fields in its neighborhood, resulting from the boundary layer separation and
vortex-shedding. In particular, the mean and instantaneous primary and secondary flow structures downstream the separation point at the leading edge are scrutinized. The analysis allows to better define the homogeneous partition of the side-surface previously proposed.

2. FLOW MODELLING AND COMPUTATIONAL APPROACH

The 3D, turbulent, unsteady flow around the cylinder is modelled in the frame of the Large Eddy Simulation approach to turbulence using the classical time-dependent filtered Navier-Stokes equations

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \nu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) + \tau_{ij}^s \right),
\]

where \(x\) and \(t\) are the space and time coordinates, \(\bar{u}\) and \(\bar{p}\) are the filtered velocity and pressure, \(\nu\) is the kinematic viscosity and \(\rho\) the fluid density. The sub-grid stress tensor is expressed according to Boussinesq's assumption as

\[
\tau_{ij}^s = \nu_s \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right),
\]

so that the equation system can be closed by a transport equation for the kinetic energy \(k_i\) of the unresolved stresses (Yoshizawa 1986)

\[
\frac{\partial k_i}{\partial t} + \frac{\partial (\bar{u}_j k_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( (\nu + \nu_t) \frac{\partial k_i}{\partial x_j} \right) + P_k - C_k \frac{k_i^{3/2}}{l_e},
\]

where \(P_k = 2\nu \overline{s_{ij} s_{ij}}\), \(\nu_t = C_k l_e k_i^{1/2}\), the constants are set equal to \(C_k=1.05\), \(C_k=0.07\) and \(l_e=l_k=\Delta\), and \(\Delta\) is the characteristic spatial length of the filter, related to the mesh size and defined as the cubic root of the mesh cell volume. The modelling of the flow in the turbulent boundary layer is accomplished by introducing a filter width \(\delta\) damped according to the Van Driest approach:

\[
\delta = \min \left[ \Delta, \frac{k}{C_A} y \left( 1 - \exp \left( -\frac{y^*}{A^*} \right) \right) \right],
\]

where \(k=0.4187\) is the Von Karman constant, \(C_d=0.158\), \(A^*=26\) is the Van Driest constant, \(y\) the distance to the wall, \(y^*=u_r y/\nu\) the non dimensional wall unit and \(u_r\) the shear velocity (Mason and Thomas 1992, De Villiers 2006).

The computational domain and the boundary conditions are shown in Figure 1. The spanwise length of the computational domain is set equal to \(L/B = 1\) on the basis of a short review of the state of art. On the basis of several studies addressed to the case of the square cylinder, Tamura et al. (1998) have proposed a minimum requirement for the spanwise length \((L/B \geq 1)\) where the breadth \(B\) of the bluff cylinder is given as a reference length. To the authors' knowledge, detailed experimental data on the spanwise correlation length for low-degree of bluffness cylinders are not available. Other LES simulations applied to more elongated, reattached-type rectangular cross-sections have generally adopted spanwise length below or equal the lower bound proposed by Tamura et al. (1998): for instance, in Yu & Kareem (1998) \(L/B=0.5\) \((B/D=4)\), in Mannini et al.

Dirichlet conditions on the velocity field and on the sub-grid kinetic energy are imposed at the inlet boundaries. Neumann conditions on the normal component of the stress tensor $\Gamma$ and on $k_t$, are imposed at the outlet boundaries. Periodic conditions are imposed on both the side surfaces and on the upper-lower surfaces, as depicted in Figure 1. No-slip conditions are imposed at the section surface. The initial conditions are obtained from a previous LES simulation, where the standard Smagorinsky sub-grid model (Smagorinsky 1963) was employed.

A hexahedral grid is adopted to discretise the spatial computational domain. The grid is hybrid in the $x$-$y$ plane and structured along the spanwise direction $z$. The grid in the $x$-$y$ plane is shown in Figure 2 with a close-up view around the leading edge.

A body-fitted, structured grid boundary layer is generated at the wall, where the constant grid spacing normal to the cylinder wall is $\Delta y/B = 5.e-4$. An unstructured quadrilateral grid is used in the remaining part of the $x$-$y$ plane to obtain an effective cell distribution on the basis of the expected flow phenomena to be simulated. The 3D grid is obtained by the structured projection of the 2D hybrid grid along the spanwise direction $z$, where 24 cells are employed to uniformly discretise the domain length $L=B$. The adopted grid generation strategy permits to achieve a good enough spatial resolution with an overall number of cells (about $1.75 \times 10^6$) lower than the one required for fully structured grids, such as the Cartesian or O-type ones. The resulting non-dimensional mean wall distance value $y^+$ is close to the unit. The non-dimensional time-step needed for an accurate advancement in time is $\Delta t = 5.e-3 \ tU/D$. The simulation is extended over $T = 800 \ tU/D$ non
dimensional time units in order to overcome the transient solution and to allow the statistical analysis of the periodic flow.

The OpenFoam® Finite Volume open source code is used in the following to numerically evaluate the flow-field. The cell-centre values of the variables are interpolated at face locations using the second-order Central Difference Scheme for the diffusive terms. The convection terms are discretised by means of the so-called Limited Linear scheme, a 2nd order accurate bounded Total Variational Diminishing (TVD) scheme resulting from the application of the Sweby limiter (Sweby 1984) to the central differencing in order to enforce a monotonicity criterion. Advancement in time is accomplished by the two-step Backward Differentiation Formulæ method. The pressure-velocity coupling is achieved by means of the pressure-implicit PISO algorithm, using a predictor-corrector approach for the time discretisation of the momentum equation, whilst enforcing the continuity equation. Computations are carried out on 8 Intel Quadcore X5355 2.66GHz CPUs and require about 2.5GB of memory and 15 days of CPU time for the whole simulation.

3. APPLICATION AND RESULTS

The incoming flow is characterised by a \( \text{Re} = \frac{UD}{\nu} = 4 \times 10^4 \) Reynolds number, where \( U \) is the free stream velocity, an incidence \( \alpha = 0 \) and a turbulence intensity \( I_t = 0\% \) (ideal smooth flow). The cylinder rectangular cross section is characterised by sharp edges and smooth surfaces.

In order to focus on the 2D flow phenomena that mainly affect the aerodynamic behaviour of the cylinder, the recognised mean flow structures are pointed out and discussed first. Some unclear aspects of the mean flow downstream the separation point are clarified by analysing the flow dynamics in the same zone. Finally, an attempt is made to relate the mean field along the side surface and the instantaneous field downstream the separation point.

3.1 2D Mean Flow

The topology of the mean flow around the obstacle is shown in Figure 3: the pathlines obtained from the velocity field averaged in time and along the spanwise dimension are plotted in the upper part of the figure, while a synthetic scheme of the recognised mean flow structures is proposed in the lower part.

![Figure 3: Pathlines averaged in time and along the spanwise dimension (above) and scheme of the recognised mean flow structures (below)](image)

The mean flow separates at the leading edge and reattaches just upstream the trailing edge, while the reversed flow in the wake approximatively extends along \( 0.76 \) \( D \). The main vortex shows an inclined major axis, while a thin recirculation region is clearly visible close to the lateral wall, between the main vortex and the separation point, without reaching the latter. In a large flow region between the main vortex and the recirculation region, called herein “inner region”, no mean
structures can be easily recognised.

![Figure 4: Recognised mean flow structures and friction coefficient distribution](image)

The mean wall shear stress coefficient $\overline{C_f}$ distribution on the lower half perimeter is plotted in Figure 4 and ascribed to the mean flow structures discussed above. On one hand, the changes in sign of $\overline{C_f}$ permit the $x$-length of these structures to be measured. In particular, the mean reattachment point can be located as the point between the counter-clockwise main vortex, along which $\overline{C_f} < 0$, and the reattached flow, along which $\overline{C_f} > 0$. It follows that the distance of the reattachment point from the separation one is equal to $x_R/B = 0.933$, which is larger than the one estimated by Matsumoto et al. (2003) based on the distributions of the time-averaged pressure coefficient and rms value ($x_R/B = 7/8$). This slight discrepancy can be ascribed to the adopted different identification methods, but another possible explication can be found looking at the difference between the present incoming flow conditions (ideal smooth flow) and the experimental ones affected by wind tunnel residual incoming turbulence (Laneville and Williams 1979, Nakamura and Ozono 1987, Bruno and Fransos 2008). Deeper sensitivity studies would be required to verify this hypothesis. On the other hand, the $\overline{C_f}$ distribution allows to shed some light into the "inner region" flow. In fact, the clockwise recirculation region involves positive $\overline{C_f}$ values, while negative ones are located just downstream the separation point up to the recirculation region. According to the authors, this second interval cannot be directly ascribed to an upwind extension of the main vortex, but could correspond to other counter-clockwise flow structures. Bearing in mind that POD analysis (Bruno et al. 2008) has shown that the flow is mainly 2D along the inner region, time-averaging is supposed to partially hide the instantaneous structures which take place in this region during the vortex growth and shedding.

### 3.2 Flow dynamics downstream the separation point

In order to verify the hypothesis above, the dynamics of the local flow downstream the separation point is analysed in the following. Figure 5 shows the instantaneous pathlines in this region: the patterns are sampled with a step equal to 0.25 non dimensional time unit during a sampling windows equal to 4.25 non dimensional time units, corresponding to half a period of the fluctuating lift force, approximatively.
Some recognised clockwise and counter-clockwise vortices are sketched. Dashed lines joint the vortex centre in successive times. The sequence mainly takes into exam the shedding process of three vortices \(v_1, v_2, v_3\) and the related flow structures. Grey dash-dot lines refers to vortices successively shed. The qualitative exam of the sequence does not aim to provide the rigorous measure of the flow structures, but to contribute to understanding the essential physics of the local flow. The vortices are not shed from the separation point, but from the apex of a pseudo-triangular region just downstream it (outlined for instance by dash lines at times since 787.25 to 788.50) which includes a sort of elongated clockwise “bubble” in the shear layer just downstream separation together with a secondary counter-clockwise vortex close to the surface, caused by the velocity field induced by the growing vortex. The pseudo-triangular region remains substantially attached to the body during the shedding cycle, even if it oscillates and its geometry changes: the secondary vortex is nearly constant in time and space, while the “bubble” slightly pulsates attaining its maximum and
minimum elongation just before and after the vortex shedding, respectively. These structures, which characterise the simulated fully-developed flow downstream the separation point, recall the ones first recognised by Pullin and Perry (1980) in the transient flow involved by a starting vortex past a 90° edge: for this reason, the same nomenclature as in Pullin and Perry (1980) is adopted.

The simulated instantaneous flow pattern at the first sampling time \( tU/D = 787.25 \) is compared with one of the flow visualisations proposed by Pullin and Perry in Figure 6: the figures are scaled in order to obtain the same distance of the apex from the wall; the Pullin and Perry visualisation is rotated with respect to the original one in order to make the comparison easier. Despite the differences between the overall flow conditions, the clear analogy seems to confirm the research perspectives expressed by Buresti (1998): “[…] it is reasonable to infer that many of the features observed in the transient flow field around the wedge, induced by increasing the upstream velocity, may be qualitative similar to those occurring near a bluff body separation point during the roll up of a forming vortex”. In other terms, the pseudo-triangular region seems to be a case-insensitive, basic flow structure in bluff-body aerodynamics, even if its “extent is a function of the body shape, and in particular of the afterbody” (Buresti 1998, see Braza et al. 1986 for the circular cylinder). In the present case-study, the pseudo-triangular region spans over about 5/8D from the separation point.

The primary vortex shedding period is equal to about 0.75 non dimensional time unit and it is convected along the wall with a velocity estimated around of 0.48U. At \( tU/D = 788.75 \), three primary vortices \( (v_1, v_2, v_3) \) successively shed from the apex are travelling along the side surface. During the following non-dimensional time unit (second column in Figure 13), they successively coalesce in a single vortex \( (v_{1,2,3}, \ tU/D = 790.00) \). The resulting vortex induces a secondary counter-clockwise vortex \( \hat{v} \) close to the side surface. Unlike the nearly-constant secondary vortex included in the pseudo-triangular region, the vortex \( \hat{v} \) is convected upstream by the velocity field induced by the main vortex with a velocity estimated around of 0.14U.

### 3.3 The lateral surface mapping revisited

Once the mean field along the side surface and the instantaneous field downstream the separation point have been discussed, Figure 7 (a) makes an attempt to relate them: the instantaneous pathlines
refer to the sampled time \((tU/D=788.75)\) at which the maximum number of primary vortices \((v_1, v_2, v_3)\) are present at the same time past the pseudo-triangular region; the recognised instantaneous structures in the mean inner region are plotted with continuous lines, while the mean structures are drawn with dashed lines.

![Diagram](image)

Figure 7: Instantaneous pathlines, flow structures and lateral surface mapping (a), pressure coefficient distributions along the central section (b-d)

It follows that the mean inner region can be viewed as the one which contains the pseudo-triangular region, the primary vortices shed from it and the secondary counter-clockwise vortices induced by the large vortex resulting from the coalescence of the primary vortices. In particular, the mean recirculation region results from the contribution of several instantaneous vortices, namely the steady one included in the pseudo-triangular region and the ones convected upstream. Figures 7 (b)-
(d) graphs the distributions of the pressure mean value, standard deviation and skewness, respectively. Both the recognised flow structures and the pressure distributions allow to better define the guess, physical-based mapping of the lateral surface firstly introduced in Bruno et al. (2008). The mapping results from four recognised zones, whose extent is roughly evaluated at the external boundary of the separating shear flow: they are named, quoted and graphically represented with grey patterns in Figure 7. It is worth stressing that the rigorous identification of the $x$-length of these zones, even though possible, is not the scope of this work, while approximate but phenomenon-based lengths have been preferred to make a guess at the relationship between the fluid flow phenomena and the pressure distribution along the side surface.

The “vortex shedding” ($vs$) region is defined as the $x$-distance from the separation point to the apex of the mean inner region: it contains the instantaneous primary vortices shed by the pseudo-triangular region and it is characterised by a $C_p$ plateau and low $C_p$ values, due to the quasi-steady behaviour of the pseudo-triangular region. The main vortex $x$-length is split into two zones in order to distinguish the part of the side surface where the coalescence of the primary vortices takes place and the one where the instantaneous reattachment occurs: the watershed point between these regions is obtained not only by looking at the point where the vortex-induced reversed flow close to the wall has a non null vertical component, but also remembering the critical aspect ratio $B/D \approx 3$ that distinguishes separated-type and reattached-type rectangular sections. The “vortex-coalescing” ($vc$) zone shows the maximum value of the mean suction and a steep increase of the fluctuating component, while the mean “pressure recovery” gives the name to the second region ($pr$), where the maximum rms value also occurs. The “mean reattachment flow” ($rf$) region is regained from above and it is characterised by another $C_p$ plateau. It is worth pointing out that the longest lengths show a change in sign of the pressure skewness and that the bound of each length corresponds to its relative maximum or minimum values. Although deeper studies are needed to interpret this evidence, the features of the $C_{p,sk}$ distribution seems to confirm the significance of the selected zones.

Finally, the regions are characterised looking at the spectral content of the pressure coefficient.

![Figure 8: Pressure coefficient time histories and PSD along the upper side surface](image)

The pressure coefficient evolution in time at the mid point of each length and the normalised
Power Spectral Densities (PSDs) of the pressure are related to the regions in Figure 8. The pressure fluctuations in the $vs$ and $pr$ lengths are mainly characterised by one frequency component, which corresponds to the prevailing frequency in the lift coefficient (Strouhal number $St$). On the contrary, the points in the $vc$ and $rf$ lengths show a broad band spectrum, where the most significant frequencies are higher than $St$, in particular two and three times the Strouhal number. Further studies are required to provide a physical-based interpretation of the frequency content of the pressure coefficient in the $vc$ and $pr$ regions, on the basis of the flow dynamics discussed in this paper (Fig. 5-7) and of new analysis of the flow close to the trailing edge, respectively.

4. CONCLUSIONS

A computational study has been proposed in this work to analyse some flow features of the high Reynolds number, turbulent, separated and reattached flow around a fixed rectangular cylinder with chord-to-depth ratio equal to 5.

The computational approach postprocessing facilities have been employed to shed some light on the relationships between the vortex shedding mechanisms and the mean and instantaneous pressure field along the cylinder lateral surface. In particular, homogeneous regions along the side surfaces have been proposed and the so-called “mean pressure recovery” region has been identified as the one that gives the most significant contribution to the lift force.

Further studies are required to check the present proposal, to complete the knowledge of the main fluid flow phenomena which drive the section aerodynamics and to provide a complete database for validation and comparison purposes.

In this perspective, the BARC benchmark could offer an useful research framework to the scientific community adopting both computational and experimental approaches.

REFERENCES

Ercoftac Test Case LES2, http://ercoftac.mech.surrey.ac.uk/LESig/les2/homepage.html, Ercoftac Database Case 43.
surface pressure field on 2D bluff bodies”, Journal of Wind Engineering and Industrial Aerodynamics, 91, 155–163.