The role of inflow boundary layer thickness for numerical modelling of flow around bluff bodies of built-up areas

Renata Gnatowska[1,*]

[1]Czestochowa University of Technology, Faculty of Mechanical Engineering and Computer Science, 21 Armii Krajowej Av, 42-200 Czestochowa, Poland

Abstract. The paper presents the numerical results obtained with the use of the ANSYS FLUENT commercial code for analysing the flow structure around a two rectangles in line surface-mounted bluff bodies immersed in a boundary layer. The simulation of a configuration with a flow around objects has been done using both the steady and unsteady approaches. Effects of the inflow boundary layer thickness on the flow field, wall-shear stress and the grid resolution accuracy in predicting periodic vortex shedding from two tandem arrangement bodies are examined. It has been observed that the periodic vortex shedding is considerably reduced with an increase in the δ/H parameter.

Keywords: boundary layer thickness, CFD, fluid flow, rectangular bluff bodies

1 Introduction

Increased interest in the problem of bluff-body aerodynamics has been observed over the past few decades. It is governed by potential applications of the results in the design process of new buildings as well as optimization of the existing urban areas and city centers. The local wind climate influences the comfort around and between buildings, the life quality in urban areas as well as specific economic aspects of utilization of the defined zones. The flow structure of a build environment is usually studied based on simplified configurations starting from single rectangular obstacles, where a horse-shoe vortex, which extends downstream along to the obstacle side. The flow separation from the obstacle walls results in strong shear layers along which the turbulence production is high. The resulting high turbulence level increases diffusion and enhances entrainment by a shear layer of a low momentum reverse flow in the near-wake which strongly affects the local pressure gradients and increases the mixing [1]. Another often investigated configuration are cubic obstacles in a tandem arrangement, where the flow field is further complicated by mutual interference. For that case, a small separation (intermittent), lock-on and large separation regimes similar to the two-dimensional geometries are observed which are strongly affected by the influence of the ground vicinity [2]. The flow structure around surface-mounted obstacles depends on various factors. One of the most important factors is the oncoming

* Corresponding author: gnatowska@imc.pcz.pl

Reviewers: *Milan* Žmindák, *Richard* Pastirčák

boundary layer thickness δ related to the height of obstacles H. For thin boundary layers (δ/H<0.3) the structure of the upstream separation is characterized by a multiple secondary recirculation upstream of a horseshoe vortex [3]. For thick boundary layers (δ/H>0.7) the dynamic behavior of the pressure and velocity field is bimodal [4, 5]. Other important factors are also the oncoming flow turbulence, the non-dimensional distance ratio (S/H) and the height ratio of the two consecutive bodies. The sheared flow effect caused by the atmosphere boundary layer modifies the flow pattern on the windward side of a high-rise block. Tall buildings, defined as those which protrude above their neighbours, act as scoops to collect the wind over much of their height and deflect it to the ground level. In front of a building exposed to the wind, a vortex forms below the stagnation point. This vortex is shed on each side of the building and it can produce excessively high winds at ground level, causing discomfort or danger [6].

The studies of the wind environment around buildings are generally performed as experiments in wind tunnel. The alternative to this is the application of Computational Fluid Dynamics (CFD), which has been increasingly exploited in various ways recently. The usually applied method for computation of turbulent flows in wind engineering is the Reynolds Averaged Navier-Stokes (RANS) approach. Within this approach, the equations are averaged in time over all the turbulent scales, to directly yield a statistically steady solution of the flow variables. Another option is the use of standard turbulence models from the RANS approach in time-dependent simulations, unsteady RANS (uRANS). This approach should be applied for flows where the unsteadiness is determinable, i.e. the frequency spectrum shows a spike at a shedding frequency. This is the case of the flow around surface-mounted cubes which is analyzed in the paper.

2 Computational details

The flow around two rectangular in-line surface-mounted square cylinders immersed in a boundary layer is computed in this work. The geometry of the analysed cases are sketched in Figure 1. The calculations were carried out for configuration of two elements with different height, aligned in one line and the distance between them was S/D=2.5. The results presented in this work relate to a fixed ratio of object height H_1/H_2=0.6 and value of their "immersion" in boundary layer H_2/δ = 0.3÷1. Three-dimensional steady and unsteady RANS simulations have been carried out using a commercial CFD code, FLUENT v.17.2, with the RNG version of a ε–k turbulence model. According to the literature [7-9] this model is widely used for flows in a build environment. Users of this model have found a good agreement between computations and measurements. The spatial derivatives have been discretized using a second order upwind scheme while the pressure-velocity coupling has been achieved using the SIMPLE algorithm. The unsteady calculations have been performed using time step based on an estimation through experimental identification of dominant frequency and comprises 1% of the flow oscillations period. The blockage ratio, defined as the ratio of the frontal area of the body to the computational domain cross-sectional area has been 0.7%. For computation purposes, the flow domain is divided into a number of hexahedral cells. The mesh is non-uniform in all the three directions. The grid is clustered near the object and the spacing is increased to a proper ratio of 1.2 away from the object surface. The first cell adjacent to the walls has been set with respect to the criteria required for the individual near-wall treatment. Hence, using a two-layer approach, the width of the near-wall cell has been 0.003H, which corresponds to $1 < y+ < 3$ where H is the object height. According to the literature [10] at least 10 cells per the cube root of the object volume should be used for a flow around surface-mounted obstacles. This recommendation given above has determined the initial minimum grid resolution which is 174x74x48 per cube. The grid independent solution has been obtained for 298x130x70 by systematically refining the entire mesh in each direction, increasing the number of nodes by about 50% [5, 6, 11]. The inflow boundary layer has been prescribed according to methods proposed by Richards et al. [8, 9]. The inlet flow profile have been approximated using the power law distribution *U(z)* and also distributions of the turbulent kinetic energy k and dissipation rate ε (values are compared with experimental data) [5, 6].

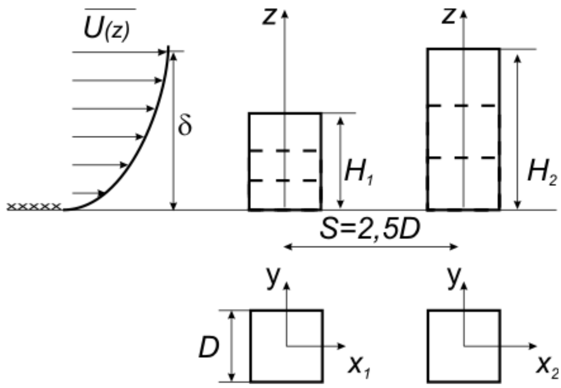

Fig. 1. The geometry of the analysed case

3 The role of inflow boundary layer thickness

The ratio of the objects height to the incoming boundary layer thickness (H/δ) has an important influence on the flow structure, and particularly, on the separation regions upstream and downstream the obstacle. As has been shown, for example, by Hunt et al. [12] and Castro and Robins [13], the shape and form of the separation region is markedly reduced by the presence of upstream near wall turbulence.

To confirm these observations, computational modelling of a flow around a multiple bluff bodies arrays immersed in a thin and thick boundary layer has been performed. The thin boundary layer is equal to the height of second cube ($H_2/\delta = 1$) while the thick boundary layer is three times higher than H_2 ($H_2/\delta = 0.3$). The flow around configuration of two rectangular tandem arrangement buildings, described by the distance $S/D = 2.5$ and the height ratio $H_1/H_2 = 0.6$, when the objects are arranged in the flow direction with the smaller one located on the upwind side is analysed. A strong downwash effect is observed for the relatively small spacing considered here. The big standing vortex which stabilizes the flow in the spacing does not allow the separated shear layers from the lateral sides of windward object to interact in between the obstacles. Additionally, it produces an excessively high velocity at the ground level what is directly correlated with the wall shear stresses distribution on the ground. It is worth mentioning that the downwash effect becomes weaker and the shear layers manage to curl in the gap [11] for larger spacing values not considered in this paper. The simulation of a configuration with a flow around objects has been done using both the steady (RANS) and unsteady (uRANS) approaches.

The wall shear stress (τ [Pa]) distribution at the symmetric line ($y/D = 0$) and at the line $y/D = 0.5$, which is located along the upstream object edge extension in the gap between objects for three flow cases is presented in Figure 2. It can be observed that the size of the cavity region is significantly reduced in the presence of thicker boundary layer. The maximum amplitude of shear stresses in the cavity zone range is also smaller. Similar conclusion has been formulated by Iaccarino et al. [14] who show the difficulty in reproducing the main characteristics of the flow. Oscillations are a result of coupled oscillations between the separated shear layers from lateral sides, which is similar to the vortex shedding process for two-dimensional obstacles. But for surface-mounted obstacles this process is modified by the shear layer along the obstacle free end (top), the oncoming flow shear gradient and horseshoe vortices. In that case using the uRANS calculations should improve the agreement between the numerical and experiment prediction of the cavity zone.

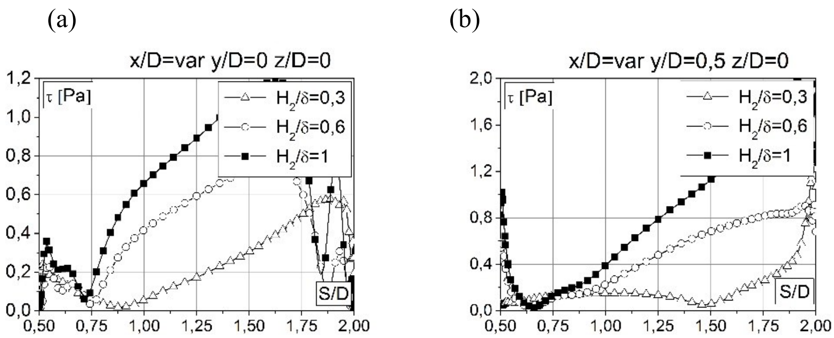

Fig. 2. Distribution of wall shear stress for steady simulation (H_2/δ=0.3; H_2/δ=0.6; H_2/δ=1.0) in the gap between objects: (a) at the line y/D=0; (b) at the line y/D=0.5

A small discrepancy between steady and unsteady results is recognized at the symmetric line (y/D=0). It is possible as the flow in the spacing is stabilized by the downwash effect. Strong differences between the results are observed at the line y/D=0.5. In this plane, the separated shear layers forming the windward obstacle's lateral sides cannot interact in the gap. The distribution of wall shear stresses of an unsteady simulation for three cases of H_2/δ is compared in Figure 3. In left column is presented the data taken along the central line y/D=0 and in right column along the line y/D=0.5. The analysis begins from the case of configuration totally immersed in a boundary layer H_2/δ=0.3 (first row). The time evolution of wall shear stresses on the ground over one period is observed and very week periodic motion in the gap between objects. The literature analysis, i.e. Castro [15], Sakamoto and Haniu [2], gives indication that a thick boundary layer inclines to suppress the periodic motion in the wake of a cube. The flow is periodic when the parameter is equal H_2/δ=1, what is also reported in Figure 3c. A similar numerical results has been also performed by Iaccarino et al. [14] and Huptas et al. [11], but analysis was performed for flow around a single cube immersed in a thin boundary layer, δ/H=0.07 and δ/H=0.1 respectively. The above results confirm the important role of the inlet boundary layer thickness on the flow structure around the obstacle.

Conclusion

A numerical analysis of the objects interaction in the bluff bodies arrays directly in the gap between bodies reveals the importance of the inlet boundary layer thickness on the flow structure, wind comfort around buildings [5, 16]. It has been shown that the intensification of the flow oscillation is significantly promotes with a decrease in the value of the relationship between the thickness of the layer and the height of the objects (δ/H). It is apparent especially in the surroundings of the second element. This indicates the need to use unsteady methods of flow modelling around objects.

Obtained from the non-stationary calculations (uRANS) averaged values of the dependent variable do not differ significantly from the stationary results (RANS). The results obtained from the uRANS methodology show better compliance with the experiment when the flow is not statistically stationary (H/δ>1). In flow with a thick boundary layer (H/δ<1), for non-stationary calculations, although the experimental frequency identification, do not recognize any oscillation. The cause may be a much stronger attenuation of flow through the layer in the case of numerical analysis.

(a)
(b)
(c)

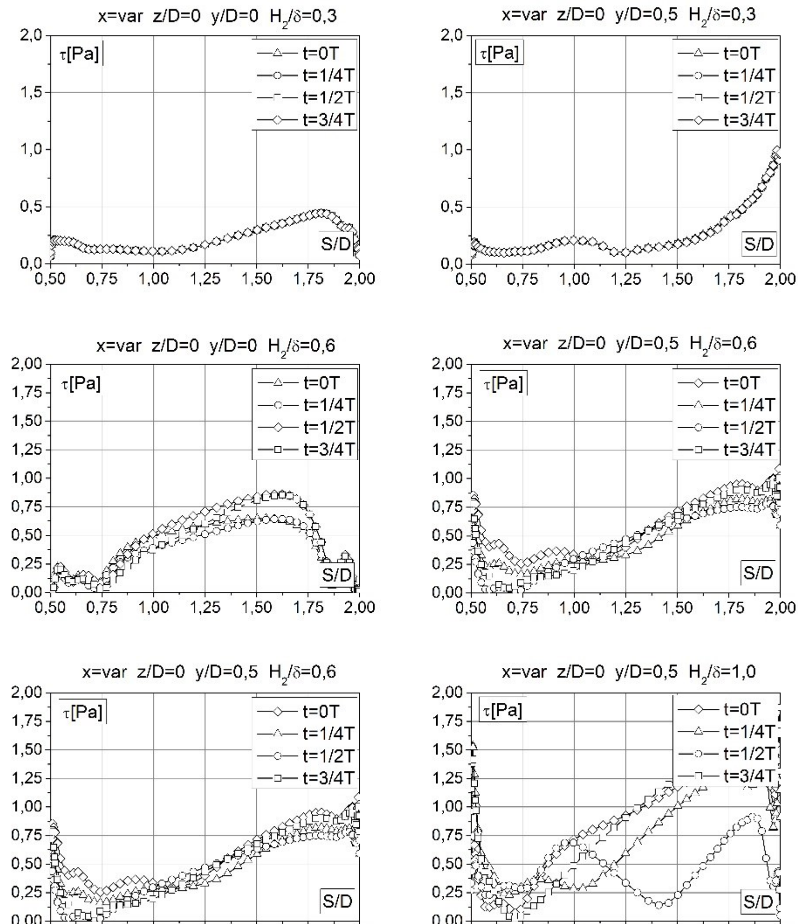

Fig. 3. Distribution of wall shear stress for unsteady simulation in the gap between objects at the line y/D=0 (left column) and at the line y/D=0.5 (right column) for (a) H_2/δ=0.3; (b) H_2/δ=0.6; (c) H_2/δ=1.0

The work was partially supported by National Science Centre, Poland (Narodowe Centrum Nauki) under grant number NCN 2017/01/X/ST8/00076.

References

1. R. Martinuzzi, C. Tropea, *The flow around surface-mounted, prismatic obstacles placed in a fully developed channel flow*. ATJFE **115**, 85-85 (1993)

2. H. Sakamoto, H. Haniu, *Aerodynamic forces acting on two square prisms placed vertically in a turbulent boundary layer*. J. Wind Eng. Ind. Aerodyn. **31** (1), 41-66 (1988)
3. E. Logan, W.H. Schofield, *Turbulent shear flow over surface mounted obstacles*. Int. J Heat Mass Transfer **112**, 376-385 (1990)
4. R.J. Martinuzzi, B. Havel, *Turbulent flow around two interfering surface-mounted cubic obstacles in tandem arrangement*. J. Fluids Eng. **122** (1), 24-31 (2000)
5. R. Gnatowska, *Aerodynamic Characteristics of Three-Dimensional Surface-Mounted Objects in Tandem Arrangement* Int. J. Turbo Jet. Eng. **28** (1), 21-29 (2011)
6. R. Gnatowska, M. Sosnowski, V. Uruba, *CFD modelling and PIV experimental validation of flow fields in urban environments*. In E3S Web of Conferences **14**, 01034 (EDP Sciences 2017)
7. A. D. Ferreira, A. C. M. Sousa, D. X. Viegas, *Prediction of building interference effects on pedestrian level comfort.* J. Wind Eng. Ind. Aerodyn. **90** (4), 305-319 (2002)
8. P. J. Richards, R. P. Hoxey, *Appropriate boundary conditions for computational wind engineering models using the k-ϵ turbulence model.* J. Wind Eng. Ind. Aerodyn. **46**, 145-153 (1993)
9. P. J. Richards, G. D. Mallinson, D. McMillan, Y. F. Li, *Pedestrian level wind speeds in downtown Auckland*. Wind Struct. **5** (2-3-4), 151-164 (2002)
10. J. Franke, C. Hirsch, A. G. Jensen, H. W. Krüs, M. Schatzmann, P. S. Westbury, N. G. Wright, *Recommendations on the use of CFD in predicting pedestrian wind environment*. (Final report, 2004)
11. M. Huptas, W. Elsner, *Steady and Unsteady Simulation of Flow Structure of Two Surface-mounted Square Qbstacles*. Task Quarterly **12** (3), 197-207 (2008)
12. J. C. R. Hunt, C. J. Abell, J. A. Peterka, H. Woo, Kinematical studies of the flows around free or surface-mounted obstacles; applying topology to flow visualization. J. Fluid Mech. **86** (1), 179-200 (1978)
13. I. P. Castro, A. G. Robins, *The flow around a surface-mounted cube in uniform and turbulent streams*. J. Fluid Mech. **79** (2), 307-335 (1977)
14. G. Iaccarino, A. Ooi, P. A. Durbin, M. Behnia, *Reynolds averaged simulation of unsteady separated flow.* Int. J Heat Mass Transfer. **24** (2), 147-156 (2003)
15. I. P. Castro, *Measurements in shear layers separating from surface-mounted bluff bodies*. J. Wind Eng. Ind. Aerodyn. **7** (3), 253-272 (1981)
16. R. Gnatowska, *A Study of Downwash Effects on Flow and Dispersion Processes around Buildings in Tandem Arrangement*. Pol. J. Environ. Stud. **24** (4), (2015)