IV Simpósio Paranaense de Modelagem, Simulação e Controle de Processos ISSN : 1984-7521	Artigo: 10
	Páginas: 67 - 74

NUMERICAL STUDY OF THE CROSSFLOW OVER DIFFERENT BODY GEOMETRIES

Matheus R. Barbieri, Bárbara L. da Silva, Jaqueline Catapan, Raissa S. Lazzaris,

Jonathan Utzig, Henry F. Meier*

University of Blumenau – FURB, Blumenau – SC, meier@furb.br

Abstract – The flow around bluff bodies has received great attention due to its known importance in practical applications. While the crossflow over circular cylinders has been widely investigated in the past years, the study with other prismatic bodies still presents gaps due to their sharp edges, that affect the wake formation and fluid dynamic forces. In this context, this study investigates the flow around triangular and square cylinders under different angles of attack through numerical simulation, in order to evaluate their wake pattern, force coefficients and frequencies. The flow pattern developed by Triangle 0° is like the circular cylinder's one, resulting in the same Strouhal number and comparable force coefficients statistics. Square 0° and Triangle 180° showed a similar behavior between them, which are the opposite of the circular cylinder, presenting an anticipated separation of the flow and greater velocity fluctuations. The Square 45° showed an intermediate behavior.

Keywords: Flow-induced vibrations. Large Eddy Simulation. Flow patterns. Force coefficients. Different geometries bodies.

Introduction

The crossflow over bluff bodies has received special attention for the understanding of its fluid dynamics, motivated by physical situations that range from civil construction to industrial equipments. From an engineering point of view, the study of whether a structure has potential to experience or not damaging vibrations is of fundamental interest. These bodies normally create a large region of separated flow and an unsteady wake region downstream, resulting in several different local instabilities, which can lead to global instabilities [3].

Although great effort has been focused in cylindrical bodies, there is still a lack of knowledge on the flow past other prismatic bodies. The separation points for the flow past bluff bodies with sharp edges remain fixed at the said edges, affecting shear layer development, wake formation and fluid dynamic forces. At low Reynolds numbers, the flow around a rectangular cylinder separates at the trailing instead of the leading edges, in a way that the separation is imperceptibly expecting to immediately reattach. By increasing the Reynolds number, the separation starts at the leading edges and steady reattachment occurs no more [1].

Some numerical and experimental studies have been conducted focusing on the evaluation of the flow characteristics around rectangular-section cylinders. The tridimensional solution of turbulent flows past bluff bodies usually requires approaches of higher computational cost, such as the Large Eddy Simulation (LES). Therefore, simplifications are commonly adopted in the computational domain to allow for reasonable simulation times. Silva and co-workers [7] studied a periodic and symmetry condition in the spanwise boundaries of the domain in comparison with a complete domain and experimental data. The authors concluded that a symmetry boundary condition provided an outcome close to the full domain simulation, which should be prioritized over the periodic condition simplification if computational and time resources allow for it.

14 e 15 de março de 2019
Curitiba - Paraná

One of the core parameters in rectangular-section bodies is the side ratio (SR), found to be inversely proportional to the Strouhal number. This dimensionless number is defined as $St = fD/V_{in}$, where f is the vortex shedding frequency, D is the characteristic length of the body, equal to the diameter in the case of a circular cylinder and V_{in} is the inlet velocity [3],[15]. In previous studies, the Large Eddy Simulation of the flow characteristics over rectangular cylinders with SR varied from 0.4 to 4 in a Reynolds number of 100000 [3] showed that the flow separates from the upstream corners and do not reattach onto the upper and lower faces for cases with side ratios inferior to 2.5. Through the hot-wire anemometry technique, Sarioglu and Yavuz [15] acquired experimental data of the flow around circular and rectangular cross-section cylinders, with the latter having different width-to-height ratios of 0.5, 1.0 and 2.0. The authors found that vortex shedding is directly proportional to the Reynolds number for all models tested and inversely proportional to the wake width. Besides, the Strouhal number increased with increasing blockage ratio, which is defined as the portion of flow channel that is occupied by the body.

Okajima [1] conducted experiments in a water tank and wind tunnel for various rectangular cylinders. The author analyzed how the Strouhal number varies with width-to-height ratios through measurements of the velocity distribution and flow visualization. Flow separation was observed at the leading edges, followed by its rolling-up behind the body.

Norberg [11] investigated experimentally the flow around and pressure forces on fixed rectangular cylinders at angles of attack between 0° and 90° through hot-wire measurements. Multiple frequencies were sometimes found at small angles of attack, within certain Reynolds numbers. For intermediate angles of attack with a fully separated flow, the Strouhal number and drag coefficient were approximately constant with the Reynolds number.

Vickery [4] conducted experimental measurements of fluctuating lift and drag on a long square cylinder with various angles of attack. The author analyzed the influence of both steady and turbulent stream and found a strong dependency of the force coefficients with large-scale turbulent structures. These changes included an increase in wake pressure and reduction in the fluctuating lift force of about 50%.

Bao and co-workers [18] investigated numerically the flow around prismatic cylinders using the Spalart-Allmaras turbulence model in a Reynolds number of 22000. When the cylinder shape was varied from triangle to square, the mean drag force, fluctuation of lift force and Strouhal number first increased and then decreased. The maximum values of these quantities occurred for a trapezoidal cross-section. In addition, the separated shear layer from the leading edge came closer to the side surface of the body, when the cross-section varied from triangular to square, altering the recirculation length.

The present work composes a broader study conducted by the research group at LFC/LVV – FURB, whose objective is the study of flow-induced vibration motivated mainly by industrial applications, such as heat exchangers and steam generators. In this context, Silva and co-workers [5] conducted a CFD-based design of experiments in tube bundles with water as working fluid, considering four transversal and longitudinal spacings between cylinders. Through the evaluation of flow patterns and force coefficients statistics, the authors emphasized the importance of considering different spacings between tubes once they lead to different effects over the response variables.

Silva and co-workers [9] experimentally investigated flow patterns in four bundles varying the transversal and longitudinal spacings between the cylinders. By means of the Laser Doppler Anemometry technique, a bistable biased regime was observed in cylinder arrays with small transverse spacing, causing multiple Strouhal numbers. The same authors carried out a Large Eddy Simulation [8] aiming to validate flow patterns

and velocity profiles obtained in [9]. A new classification of in-line arrangements was proposed, based on transversal and longitudinal spacings range, besides the evaluation of force coefficients acting over each cylinder.

Aiming to extend the study of flow-induced vibrations to other fluids, Barbieri and co-workers [16] conducted numerical simulations of the cross-flow over cylinder arrays with air as the working fluid. The bundles had their transversal and longitudinal spacings between tubes varied according to an experimental design, which allowed for the analysis of flow patterns and force coefficients statistics. Comparing the results with those from Silva and co-workers [5], the change of the working fluid from water to air resulted in weaker force coefficients considering the same spacings between tubes. However, the presence of two more cylinders in some arrays led to an increased mean drag coefficient and fluctuation of the lift force, even with a less dense fluid.

Accordingly, different flow patterns and force coefficients behavior are expected changing the geometry of the bluff body. Thus, a numerical study employing a Large Eddy Simulation approach is carried out aiming to contribute with the knowledge about the flow around prismatic-shaped bodies. A triangular and square cross-section cylinders are subject to crossflow with water as working fluid in a channel. Each of these bodies are varied with two angles of attack and the results are compared with a circular cylinder under the same flow conditions, as well as literature results.

Methodology and Mathematical Modeling

Some geometries were considered to extend the study of flow-induced vibration to other applications. Prisms of varied base geometries subject to crossflow with water as working fluid were simulated in a confined channel flow:

- Square base with side $L = 1$ cm and a flat face of the prism as the leading edge (Square 0°);
- Square base with side $L = 1$ cm rotated 45° in relation to the previous geometry (Square 45°);
- Equilateral triangular base with side $L = 1$ cm and two faces of the prism as the leading edges (Triangle 0°);
- Equilateral triangular base with side $L = 1$ cm and the flat face of the prism as the leading edge (Triangle 180°).

The flow around a circular base prism (cylinder) with a diameter $D = 1$ cm was taken as reference. Figure 1 presents the characteristics of the computational domain in all simulations, changing only the prism base. The height of the domain was set to $5.25D$. A uniform velocity of $V_{in} = 0.12$ m/s was set to the inlet boundary corresponding to a Reynolds number based on the cylinder diameter $Re_D = \rho V_{in} D / \mu = 1200$, while at the outlet a static pressure P was set as 0 Pa. A symmetry boundary condition was adopted at the top surface.

Hexahedral grids were generated for all cases. The closest node to the prism surfaces and the channel walls ensured maximum values of y^+ inferior to 1. The meshes have the same structure along the axis of the prism, with the maximum cell size in the vicinity of the cylinder equal to 1.25 mm, expanding up to 2.49 mm on the inlet and outlet boundaries of the domain. The size of the meshes varied between 2 and 3 million nodes.

Such grid resolution was required for the use of the Large Eddy Simulation (LES) approach, where the Navier-Stokes equations are spatially filtered for the large eddies solution. The characteristic length of the filter, that determines which scales are calculated, usually consists on the mesh size itself. Smaller scales, called subgrid scales, are modeled [10]. The Navier-Stokes equations after the mathematical filtering operation, symbolized by an upper trace over the variables, govern the flow for Large

Eddy Simulation and are written according to Eq. 1 and Eq. 2 for an incompressible flow:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0, \quad (1)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) = \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} - \frac{\partial \tau_{ij}^{LES}}{\partial x_j}, \quad (2)$$

where $\bar{u}_i$ are the filtered velocity components, x_i are the cartesian coordinates, ρ is the density of the fluid, ν is the kinematic viscosity of the fluid and $\bar{p}$ is the filtered pressure. The tensor τ_{ij}^{LES} is called subgrid-scale Reynolds tensor. It includes the effects of the subgrid scales and was modeled with the Dynamic Model proposed by Germano and co-workers [14] and modified by Lilly [12]. The decision to use the LES approach with the Dynamic Model was based on preliminary simulation tests [6].

Figure 1- Geometry for the flow simulation over prisms of diverse bases.

The finite volume method implemented in the commercial code ANSYS Fluent 14.0 with the physical properties of liquid water ($\rho = 998.2 \text{ kg/m}^3$ and $\nu = 1\text{e-}6 \text{ m}^2/\text{s}$) was used to solve the filtered three-dimensional incompressible Navier-Stokes equations for a Newtonian fluid. The pressure-velocity coupling was solved using the SIMPLEC algorithm, adopting a convergence criterion of $1\text{e-}5$ for the variables. Diffusive momentum fluxes were discretized using the Central Differencing method and convective fluxes with the Bounded Central Differencing method. A second order implicit transient formulation was used. The time steps were different for each case but ensured maximum Courant number values inferior to 1 in all cases. The drag and lift coefficients of the prisms were monitored throughout the solution of the cases, until they presented a quasi-stationary state. Then, averages were taken for at least 10 s of flow time.

Results and Discussion

The results include the evaluation of the average velocity profiles in the direction of flow (x direction, in Figure 1) and in the transverse direction (z direction, in Figure 1), temporal mean and fluctuations of the force coefficients, instantaneous velocity

contours and the Strouhal number (St), which expresses the oscillation frequency, according to Eq. 3:

$$St = \frac{fD}{V_{in}}, \quad (3)$$

where the frequency f was obtained from the application of the Fast Fourier Transform on the time history of the lift coefficient of each prism.

The velocity profiles for all simulated cases are shown in Figure 2, with the origin of the axes being in the center of the bluff body. The solid lines represent the time-average of the streamwise velocity (U/V_{in}) and crosswise velocity (W/V_{in}), while the dashed lines represent their fluctuations. The location in which the average streamwise velocity is negative indicates the recirculation region after the prism. It can be observed that the largest recirculation lengths were formed in the case of the circular base prism and the Square 45° base. According to Bao and co-workers [18], the recirculation region first decreases and then increases with the variation of the cylinder shape from triangular to square. Geometric elements with the meeting of two edges as the leading edge (Triangle 0° and Square 45°) showed more intense minimum U/V_{in} values and lower velocity fluctuations in both directions, while the elements with a flat leading edge (Square 0° and Triangle 180°) had softer minimum streamwise velocity because of the anticipated separation of the flow, and greater fluctuations of the velocity. The cases Square 45° and Triangle 180°, due to the encounter of two edges in the back of the prism (formation of a "tip"), present a sharp peak at the beginning of the U/V_{in} profiles and fluctuations of W/V_{in} .

Figure 2 - Profiles of mean velocity (solid line) and temporal fluctuation (dashed line).

Table 1 indicates the mean drag coefficient $\bar{C}_D$, the temporal fluctuation of drag coefficient C'_D , the mean lift coefficient $\bar{C}_L$, and the temporal fluctuation of lift coefficient C'_L for each prism. As a result of the flat leading edge, the cases Square 0° and Triangle 180° showed higher $\bar{C}_D$, as well as higher C'_L reflecting the fluctuation profiles of the crosswise velocity. The Triangle 0° case was closer to the circular base prism, and the Square 45° case showed intermediate values.

Although few studies were found to focus on prismatic cylinders, the values obtained in this work agree with the encountered literature results. The $\bar{C}_D$ for the Square 0° is expected to be between 2.00 and 2.20 [11],[13], while C'_L is equal 1.32 for a

smooth stream and 0.68 for a turbulent stream [4]. The Triangle 180° presents a $\bar{C}_D$ between 2.00 and 2.25 and a C'_L between 0.60 and 1.00 [17]-[18].

The formation of vortical structures can be verified through the instantaneous velocity profiles shown in Figure 3. The similarity of the wake of the circular and Triangle 0° base prisms is emphasized, which happened due to the existence of only one point of shear layer detachment, without reattachment, at each side of the prism. The other cases present more complex structures, with shear layer reattachment in the Square 45° and the Triangle 180° base prisms. In general, the formation and shedding of vortices are present in all the simulated cases, whose intensity depends on the geometry, besides of being related to the characteristics of the average velocity and force coefficient profiles.

Table 1- Temporal statistics of the force coefficients of the prisms.

Geometry of the base	$\bar{C}_D$	C'_D	$\bar{C}_L$	C'_L
Circle	1,16	0,02	0,00	0,06
Square 0°	2,61	0,15	0,04	1,23
Square 45°	2,21	0,16	-0,02	0,44
Triangle 0°	1,68	0,14	0,00	0,14
Triangle 180°	2,86	0,26	-0,03	1,11

Figure 3 - Profiles of instantaneous velocity for the simulated cases normalized by V_{in} .

Finally, the Strouhal number of each case is shown in Figure 4, corresponding to the highest peak in the spectrum. Both Cylinder and Triangle 0° cases have $St = 0.23$, an expected characteristic due to the similarity of both wakes. The other prisms showed smaller values, with Square 0° and Square 45° presenting a $St = 0.15$ and Triangle 180° a $St = 0.17$. It is observed that a second peak occurred in the simulation with triangular base prism 0°, which may be related to the formation of secondary structures with higher frequency. It was also found in the cross-flow over rectangular cylinders by [11], under small angles of attack and certain Reynolds numbers. The spectrum magnitude is proportional to the magnitude of the velocity fluctuations in Figure 2.

Vortex shedding frequency increases as the wake width decreases. In the present results, the circular cylinder presents the largest wake width, and the Square 0° presents the narrowest wake. Other studies reported that the circular cross-section cylinder showed a Strouhal number of 0.20 [15], a square cylinder with an angle of attack equal to 0° presented a Strouhal number close to 0.12 [2],[13]. Triangle 180° showed a value between 0.13 and 0.14 [18].

Figure 4 - Power spectra calculated based on the lift coefficient history of the simulated cases.

Conclusion

The crossflow over different base prism geometries was investigated through numerical simulation using a Large Eddy Simulation approach. The flow characteristics as well as force coefficients of square and triangular bodies with different angles of attack were compared to the ones of a cylindrical body. The streamwise velocity profiles showed that all geometries had shorter recirculation lengths compared to the cylinder, although Square 45° and Triangle 0° showed a more intense minimum streamwise velocity, due to a prolonged separation of the flow. These patterns are correlated with the force coefficients, where prisms with greater fluctuations of the crosswise velocity showed a more intense fluctuation of the lift coefficient, denoting a greater tendency of vibration. The temporal statistics of the force coefficients of Triangle 0° showed a great similarity with the cylinder, in a way that their vortex wake presented comparable behavior, due to the existence of only one point of shear layer detachment in each side of these geometries. Thus, these two geometries exhibited the same Strouhal number, while the other geometries showed smaller frequencies of vortex shedding in the power spectrum.

Acknowledgments

This study was financed by Coordenação de Aperfeiçoamento de Pessoal de Nível Superior – Brasil (CAPES) – Financial code 001. The authors would like to acknowledge the financial support from Petróleo Brasileiro S.A. (PETROBRAS) through the cooperation agreement 5850.0103010.16.9.

References

- [1] A. Okajima. Strouhal numbers of rectangular cylinders. Journal of Fluid Mechanics. 1982. 123, pp. 379-398.

- [2] A. Okajima; T. Nagahisa; A. Rokugoh. A numerical analysis of flow around rectangular cylinders. *JSME International Journal*. 1990, 33, pp. 702-711.
- [3] A. Sohankar. Large eddy simulation of flow past rectangular-section cylinders: Side ratio effects. *Journal of Wind Engineering and Industrial Aerodynamics*. 2008, 96, pp. 640-655.
- [4] B.J. Vickery. Fluctuating lift and drag on a long cylinder of square cross-section in a smooth and turbulent stream. *Journal of Fluid Mechanics*. 1966, 25, pp. 481-494.
- [5] B.L. Silva; J. Utzig; H.F. Meier. CFD-based design of experiments for the study of flow-induced vibrations in a tube bundle. In: *Proceeding of 10th ABCM Spring School on Transition and Turbulence*, São José dos Campos, 2016.
- [6] B.L. Silva; J. Utzig; H.F. Meier. Vibrações induzidas por escoamento: avaliação de modelos LES e RANS para escoamento cruzado sobre um corpo cilíndrico. In: *XXI Congresso Brasileiro de Engenharia Química*, Fortaleza, 2016.
- [7] B.L. Silva; M.R. Barbieri; J. Utzig; W.P. Martignoni; H.F. Meier. On the influence of spanwise boundary conditions on the Large Eddy Simulation of the flow past a single cylinder. In: *Presented at the 2017 AIChE Annual Meeting*, Minneapolis, 2017, USA.
- [8] B.L. Silva; R.D. Luciano; J. Utzig; H.F. Meier. Analysis of flow behavior and fluid forces in large cylinder bundles by numerical simulations. *International Journal of Heat and Fluid Flow*. 2019, 75, pp. 209-226.
- [9] B.L. Silva; R.D. Luciano; J. Utzig; H.F. Meier. Flow patterns and turbulence effect in large cylinder arrays. *International Journal of Heat and Fluid Flow*. 2018, 69, pp. 136-149.
- [10] C. Fureby. Towards the use of large eddy simulation in engineering. *Progress in Aerospace Sciences*. 2008, 44, pp. 381-396.
- [11] C. Norberg. Flow around rectangular cylinders: Pressure forces and wake frequencies. 1993, 49, pp. 187-196.
- [12] D.K. Lilly. A proposed modification of the Germano subgrid-scale closure method. *Physics of Fluids A*. 1992, 4, pp. 633-635.
- [13] E.D. Obasaju. An investigation of the effects of incidence on the flow around a square section cylinder. *Aeronautical Quarterly*. 1983, 34, pp. 243-259.
- [14] M. Germano; U. Piomelli; P. Moin; W.H. Cabot. A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A*. 1991, 3, pp. 1760-1765.
- [15] M. Sarioglu; T. Yavuz. Vortex shedding from circular and rectangular cylinders placed horizontally in a turbulent flow. *Turkish Journal of Engineering and Environmental Sciences*. 2000, 24, pp. 217-228.
- [16] M.R. Barbieri; B.L. Silva; J. Utzig; H.F. Meier. CFD-based study of flow-induced vibrations in cylinder arrays with air as working fluid. In: *11th Spring School on Transition and Turbulence*, Uberlândia, 2018.
- [17] P.W. Bearman; D.M. Trueman. An investigation of the flow around rectangular cylinders. *The Aeronautical Quarterly*. 1972, 23, pp. 229-237.
- [18] Y. Bao; D. Zhou; C. Huang; Q. Wu; X.-q. Chen. Numerical prediction of aerodynamic characteristics of a prismatic cylinder by finite element method with Spalart-Allmaras turbulence model. 2011, 89, pp. 325-338.