Proceedings of the 12th International Conference on Fluid Flow, Heat and Mass Transfer (FFHMT 2025)
July 15, 2025 - July 17, 2025 | Imperial College London Conference, London, United Kingdom
Paper No. 184
DOI: 10.11159/ffhmt25.184

CFD Investigation of Turbulent Cross-Flow Behavior in Tube Bundles: In-Line Vs Staggered Configuration

Yacine Kahil[1,2], Abdelkrim Benlefki[1,2], Zakaria Rahmani[1,3]
[1] Department of Civil, Mechanical and Transport Engineering, Tissemsilt University - Ahmed Ben Yahia ElWancharissi
Tissemsilt 38004, Algeria
kahil.yacine@univ-tissemsilt.dz; benlefki.abdelkrim@univ-tissemsilt.dz; rahmaniz784@gmail.com
[2] Applied Mechanics Laboratory, University of Science and Technology - Mohamed Boudiaf
Oran 31000, Algeria
[3] Laboratory of mechanical engineering, materials and structures, Tissemsilt University - Ahmed Ben Yahia ElWancharissi
Tissemsilt 38004, Algeria

Abstract - This study numerically investigates the turbulent flow around four cylinders arranged in a square configuration at a Reynolds number of 3000, focusing on understanding flow dynamics in tube bundles. Using the Reynolds-Averaged Navier-Stokes (RANS) approach with the k-ω SST turbulence model, the simulations are conducted with the open-source software Code_Saturne. Two configurations are analyzed: in-line ($\alpha = 0°$) and staggered ($\alpha = 45°$), both with a spacing ratio of P/D = 1.5. The global properties that were computed, such as the mean velocities and pressure, were discovered to be highly consistent with the available numerical measurements. The bistability phenomenon was shown based on the different fields of velocity in in-inline configuration. At the staggered arrangement, a jet flow is observed which is distinct from the regular gap flow, characterized by a narrow width and high velocity. Altering the wake shape can cause significant variations in the magnitude and direction of the jet flow, resulting in potential structural vibrations that must be considered in practical engineering applications.

***Keywords*:** CFD, Code_Saturne, turbulent flow, tube bundle, bistability.

1. Introduction

Over the last years, scientific research has made significant progress in understanding the flow around various configurations of cylinders, using both numerical simulations and experimental techniques. Studies have been conducted on the flow around one cylinder [1]-[2], where the effects of Reynolds number, angle of attack, and cylinder diameter have been investigated.

In the study of fluid flow around more than cylinder, two important parameters are the pitch-to-diameter ratio (P/D) and the alignment angle (α). P/D refers to the distance between the centers of the two cylinders divided by the diameter of the cylinder. This parameter plays a crucial role in determining the type of flow pattern that will occur, such as whether the flow is steady or unsteady, and the extent of the wake region behind the cylinders. On the other hand, the alignment angle (α) between the two cylinders and the direction of cross-flow can significantly affect the pressure distribution around the cylinders and lead to changes in the flow structure. Hence, both P/D and α are essential parameters in understanding the dynamics of fluid flow around cylinders.

In the case of two cylinders side by side, research has focused on the distance between the cylinders and the Reynolds number. A study by Afgan et al [3] utilized Dynamic Smagorinsky LES the two side-by-side cylinders were tested with varying pitch to diameter ratios at a Reynolds number of 3000. The study found that at small gap ratios, the cylinders behaved as a single bluff body with top-bottom vortex shedding, while at intermediate gap ratios, a biased flow behavior was observed with a bistable flow pattern. Finally, for higher gap ratios, the cylinders behaved much like two independent cylinders with symmetrical wake patterns and an anti-phase vortex shedding.

Tandem and staggered cylinders have also been extensively studied, with investigations into the gap between cylinders and the Reynolds number. Alam et al [4] investigated the flow characteristics and fluid forces acting on two circular cylinders arranged in various staggered configurations. They found that the lift forces mainly depended on the gap flow between the cylinders at small gap widths, regardless of the staggered angle. The maximum fluctuating drag force acting on the downstream cylinder occurred at an angle of 10 and gap widths of 2.4-3.0.

Tong et al [5] presents a numerical study of the steady flow around two circular cylinders with various arrangements at a low subcritical Reynolds number of $Re=10^3$. The study investigates the ratio P/D, which ranges from 1.5 to 4, and the angle α, which varies from 0° to 90° degrees. The pressure distribution on each cylinder changes significantly with changes in arrangement, particularly when P/D=3 and $\alpha < 60°$, resulting in movements of stagnation points and variations in pressure distribution around the cylinder surface.

Studies on three and four cylinders have been conducted in both in-line and staggered arrangements, where the effects of cylinder spacing and Reynolds number have been analyzed. Paula et al [6] They investigates experimentally the phenomenon of bistability around three cylinders in two triangular arrangements. The results identify the presence of bistable flow in the configuration where one cylinder is upstream and two are downstream, whereas the second configuration with a distinct shedding frequency does not exhibit any bistable effect. A study conducted by Pouryoussefi et al [7] examines the impact of cross-flow on the flow interference between three cylinders arranged in an equilateral-triangular pattern of equal diameter, as well as between four cylinders arranged in a square pattern, with spacing ratios P/D varying from 1.5 to 4 at subcritical Reynolds number of 6.08×104. The results show that the aerodynamic coefficients are significantly impacted by the changes in P/D. Furthermore, decreasing the value of P/D amplifies the effects of flow interference between the cylinders.

Kahil et al [3] numerically modeled the flow around four cylinders arranged in a square pattern using dynamic Smagorinsky large eddy simulation (LES). They found that the P/D = 1.5 configuration exhibited three distinguishable stable modes with biased flow, labeled as modes 1, 2, and 3.

A number of studies have begun to examine the impact of geometric parameters on the Flow pattern transitions around four cylinders arranged in an in-line square and staggered configuration [8]-][11]. The results show that the development of free shear layers around the cylinders is heavily influenced by the spacing ratio, which, in turn, has a considerable impact on the force and pressure characteristics of the four cylinders when different spacing ratios are employed. Both experimental and numerical simulations show the presence of bistable flow nature when the spacing ratio is set to P/D = 1.5.

These studies have provided insights into the complex flow phenomena that occur around multiple cylinders, which can be useful for a range of engineering applications, especially in sensitive areas such as the production of electricity through nuclear power.

The increase in computer power and the development of numerical methods have made it possible to conduct three-dimensional flow calculations in several configurations, while taking into account the effect of viscosity and turbulence. These advances have made Computational Fluid Dynamic (CFD) a more and more important tool for developing and optimizing the dimensioning of various industrial processes. Among the wide range of known calculation codes, it is very important to note that all the numerical tools used to realize and analyze the results are open sources (free numerical models), in such a way that the meshes introduced into the calculation code were generated and designed using the Gmsh [12] mesh generator, the resolution of the governing equations and the whole numerical part was done by Code_Saturne [13]. The analysis of the calculation code results was visualized and processed using ParaView [14].

This paper is a contribution to the knowledge of the passage of a flow through a tube bundle through the examination of two tubes bundles (in-line and staggered). The investigation method is based on a numerical approach (RANS). The industrial context of this work is well specified, in direct connection with the applications relating to the tube bundles of heat exchangers.

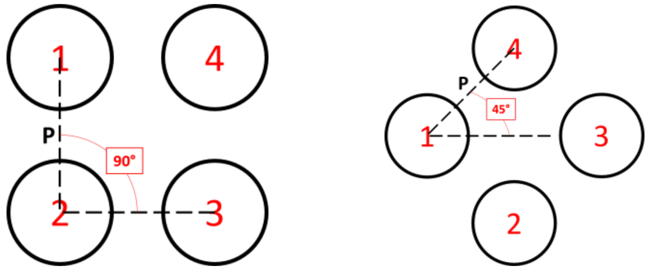


Fig. 1: Schematic of in-line (a) and staggered (b) arrangements.

2. Case setup

2.1. Computational details

Figure 2 shows the configuration of four cylinders in square arrangement with a spacing of P/D = 1.5 (measured spacing ratio between the center of the cylinders and the diameter). The compute domain dimensions are 25D*(20D+(P/D))*4D in (X), (Y) and (Z) respectively. The length upstream of the cylinder is 10D and at the entrance a uniform speed is imposed. Periodicity was applied in the (Z) direction, simulated flow in turbulent conditions for an incompressible fluid without heat transfer. The analysis is done for a Reynolds number (Re = 3000).

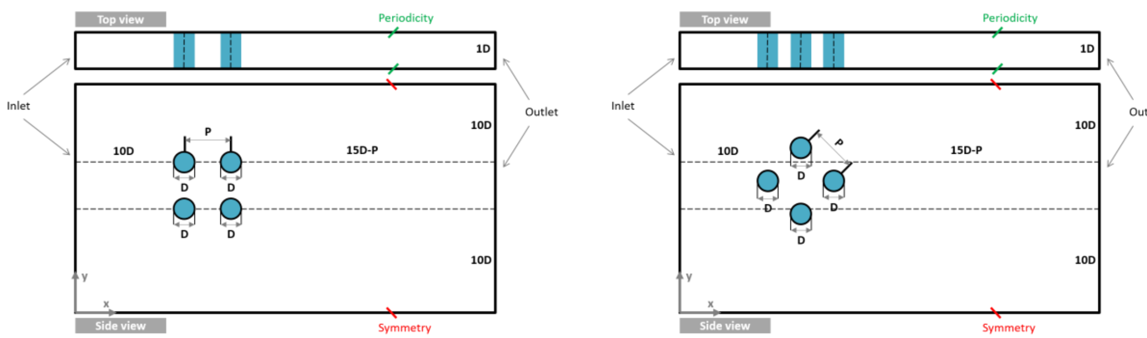


Fig. 2: Computational domain of four cylinders in a square arrangement with pitch to diameter ratio P/D = 1.5.
(a) in-line configuration, (b) staggered configuration.

2.2. Governing equations and turbulence model

Code_saturne is an open-source software created primarily by EDF to handle computational fluid dynamics (CFD) applications. It can tackle the Navier-Stokes equations for flows of 2D, 2D-axisymmetric, and 3D types, whether they are laminar or turbulent, steady or unsteady, incompressible or slightly compressible, and isothermal or not. Furthermore, it can manage scalar transport. It operates using a finite-volume technique, which can accept unstructured meshes containing a variety of cell types such as hexahedral, tetrahedral, prismatic, pyramidal, and polyhedral. While the majority of models employ cell-centered finite volumes located coherently, work is underway to implement numerical schemes utilizing CDO (compatible discrete operator).

The current work is done in the frame of Reynolds-averaged Navier–Stokes. The RANS equations are time-averaged equations of motion for fluid flow:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\rho\left[\partial \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \overline{u_i' u_j'}}{\partial x_j}\right] = \frac{-\partial \bar{p}}{\partial x_i} + \mu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} \tag{2}$$

The turbulence model used is the *k-ω SST* (Shear Stress Transport). The latter combines the original Wilcox *k-ω* model which is effective near the walls and the standard *k-ε* model effective away from the walls, using a blending function. The formulation of turbulent viscosity is modified to take into account the transport effects of turbulent shear stress.

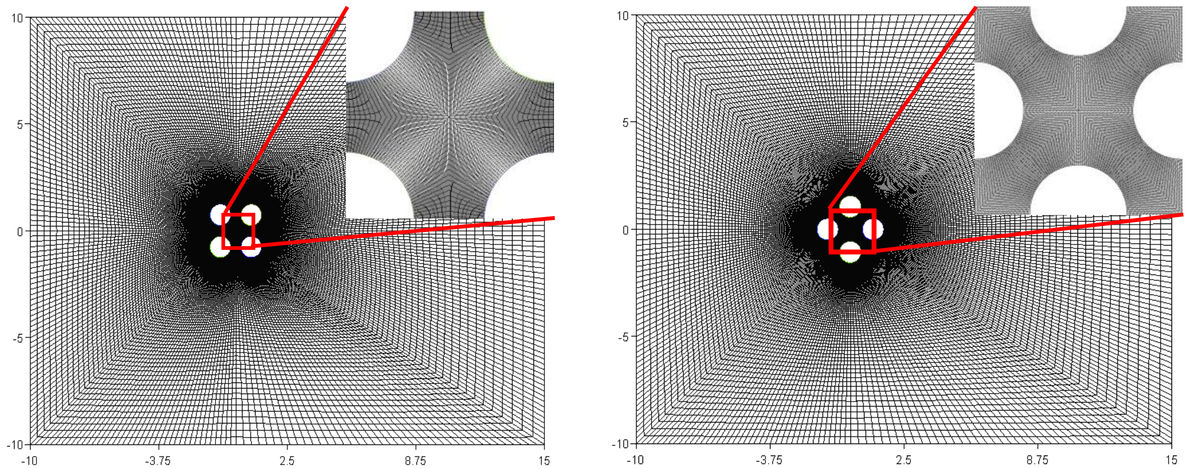


Fig. 3: Cross sectional and zoomed view of the mesh in the (XY) plane.
(a) in-line configuration, (b) staggered configuration.

2.3. Mesh dependency test

To determine the mesh sensitivity of the simulation and obtain an accurate result, the simulation run with different mesh resolutions coarse, medium, fine and very fine. Figure 4 compares the mean velocity along the centerline Y/D = 0 for both configuration in-line and staggered. From X/D = 2-4 the mean velocity vary significantly with changes in mesh resolution from coarse to fine mesh and negligible change between the fine and the very fine mesh. Therefore, the fine mesh was selected for the current study, which offers several advantages such as saving time, accurate results and requiring fewer computational resources.

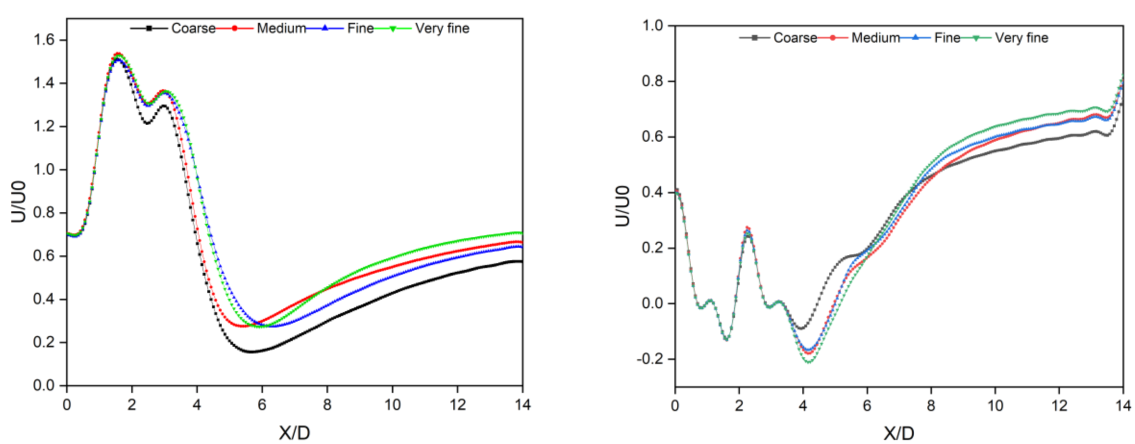


Fig. 4: Sensitivity study for the mean velocity along the centerline (Y/D = 0).
(a) in-line configuration, (b) staggered configuration.

3. Results and discussion

3.1. In-line configuration

The distribution of the non-dimensionalized mean velocity ((a) Streamwise velocity U/U0), (b) (Wall normal velocity V/U0)) in the wake of the four cylinders in in-line configuration at P/D = 1.5 is depicted in Figure 5. A comparison with the LES simulation of Kahil et al [11] at different X/D values. The agreement with the LES numerical results is good, and the only noticeable difference is observed in the Wall normal velocity V/U0 profiles at X/D = 1.5.

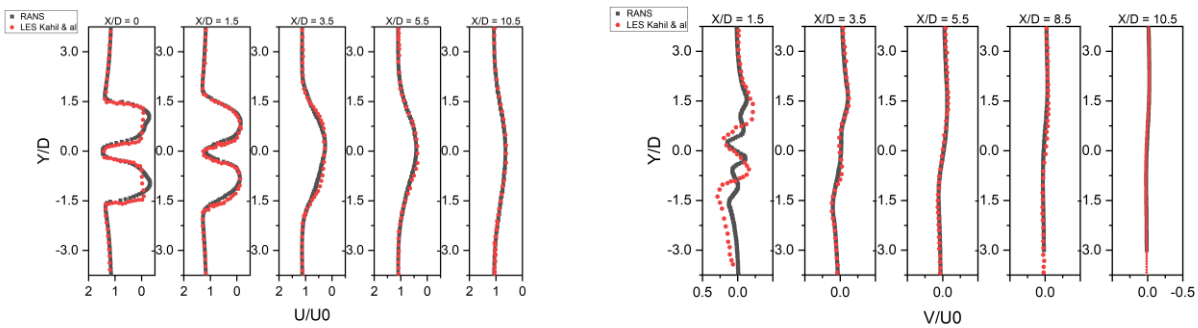


Fig. 5: Comparisons of mean velocity profiles for in-line configuration.
(a) Streamwise velocity U/U_0, (b) Wall normal velocity V/U_0.

When arranged in-line, the four-cylinder array can be viewed as either two cylinders in tandem arranged in parallel rows, or as two cylinders side-by-side arranged in parallel columns. Elongated chains of vorticity in the Kelvin-Helmholtz type of roll-up are observed in the shear layers that separate from the upstream cylinders (cylinder 1 and 2). The vortices primarily wrap around the downstream cylinders (cylinder 3 and 4) and are linked to alternate vortices that form downstream of the cylinder array (Figure 6).

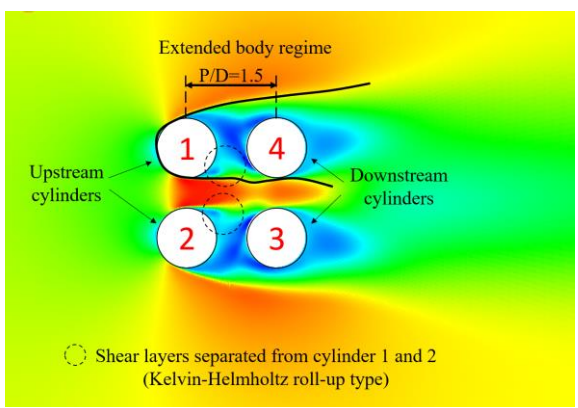


Fig. 6: The extended body regime for P/D = 1.5 showing the shear layers that separate from the upstream cylinders wrap around the downstream cylinders.

In Figure 7, the three modes (Mode 1, Mode 2, and Mode 3) are illustrated, and the wake patterns of the cylinders reveal the flow deflection with a mode change. The transition from one mode to another is non-periodic and unpredictable, which was also observed in the case of two side-by-side cylinders discussed by Afgan et al. [3]

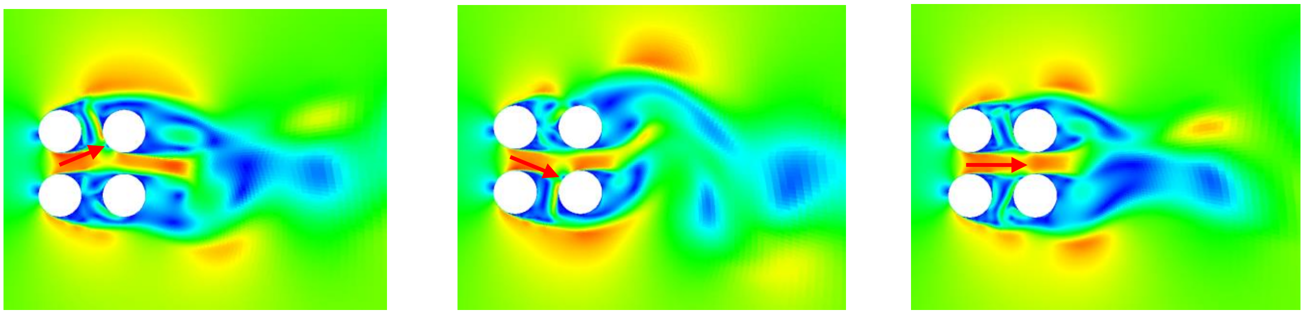
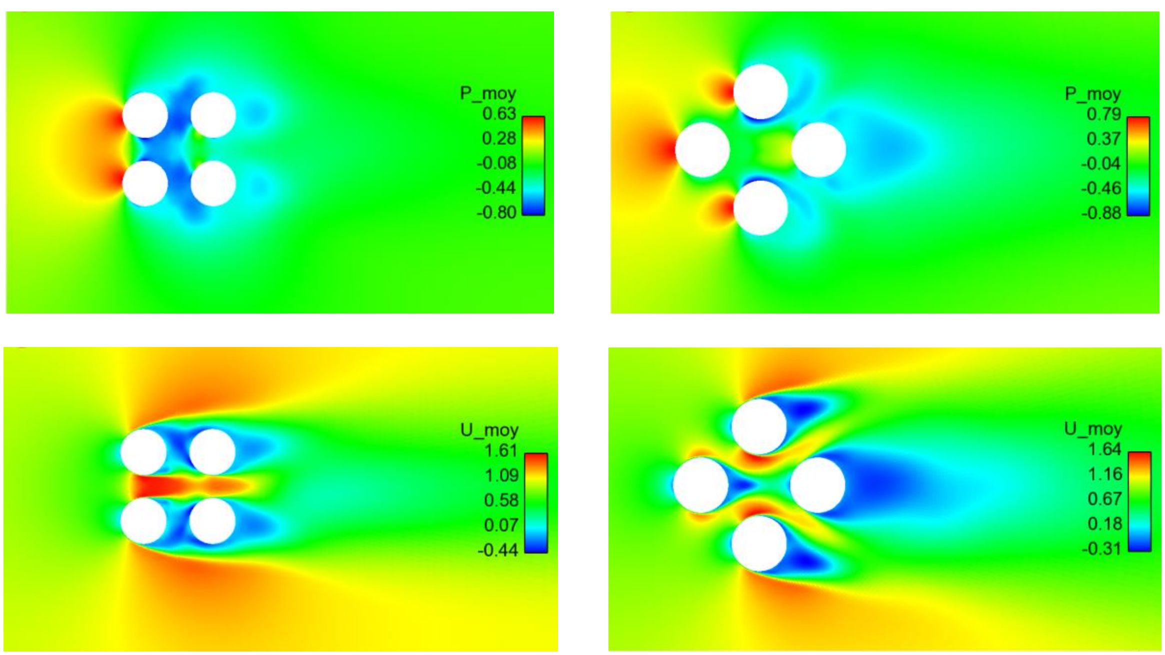


Fig. 7: Mode change (Bistability).

3.2. Staggered configuration

The current staggered configuration of the four cylinders resulted from the counter-clockwise rotation of the in-line four-cylinder by 45 degrees with the same spacing of P/D = 1.5. This arrangement places Cylinder 3 directly downstream of Cylinder 1 along the X-axis, with Cylinder 2 and Cylinder 4 positioned adjacent to each other along the Y-axis. In comparison with the in-line four-cylinder configuration, the greater level of interference between the cylinders results in a more intricate pattern of behavior being observed.

Considering the rapid variations in direction and intensity of the incoming flow onto Cylinder 3, it can be inferred that the interaction between the vortices and the cylinder structure will lead to significant fluctuations in the pressure forces and overall load on the downstream cylinder. Compared with the four cylinders in in-line configuration, the shear layers that separate from cylinder 1 are not able to extend sufficiently to encircle cylinder 3 located downstream due to the effect of cylinders 2 and 3 on the flow.



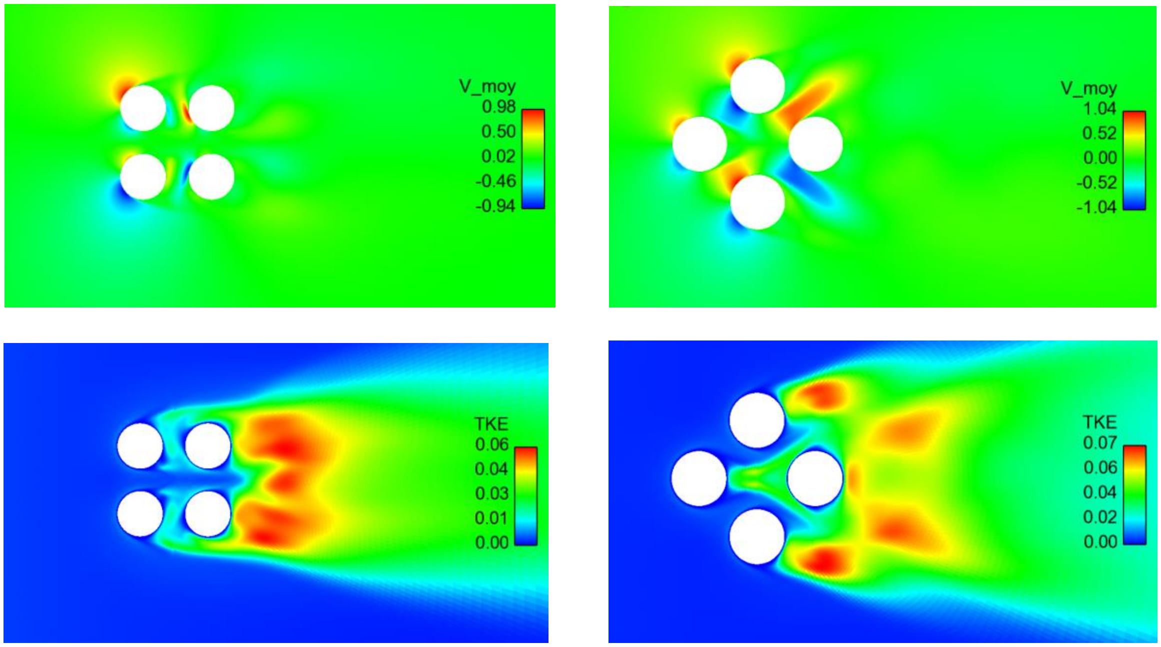


Fig. 8: Different quantities field: (a) in-line configuration, (b) staggered configuration.

4. Conclusion

At a Reynolds number of 3000, the flow around four cylinders arranged in square configurations with pitch ratios P/D=1.5 was numerically analyzed for both in-line and staggered arrangements. The global properties that were computed, such as the mean velocities and pressure, were discovered to be highly consistent with the available numerical measurements.

The bistability phenomenon was shown based on the different fields of velocity in in-inline configuration. At the staggered arrangement, a jet flow is observed which is distinct from the regular gap flow, characterized by a narrow width and high velocity. Altering the wake shape can cause significant variations in the magnitude and direction of the jet flow, resulting in potential structural vibrations that must be considered in practical engineering applications.

References

[1] P. W. Bearman and M. M. Zdravkovich, "Flow around a circular cylinder near a plane boundary," *J. Fluid Mech.*, vol. 89, no. 1, pp. 33-47, 1978.

[2] C. Norberg and B. Sunden, "Turbulence and reynolds number effects on the flow and fluid forces on a single cylinder in cross flow," *J. Fluids Struct.*, vol. 1, no. 3, pp. 337-357, 1987.

[3] I. Afgan, Y. Kahil, S. Benhamadouche, and P. Sagaut, "Large eddy simulation of the flow around single and two side-by-side cylinders at subcritical Reynolds numbers," *Phys. Fluids*, vol. 23, no. 7, p. 075101, 2011.

[4] Md. M. Alam, H. Sakamoto, and Y. Zhou, "Determination of flow configurations and fluid forces acting on two staggered circular cylinders of equal diameter in cross-flow," *J. Fluids Struct.*, vol. 21, no. 4, pp. 363-394, 2005.

[5] F. Tong, L. Cheng, and M. Zhao, "Numerical simulations of steady flow past two cylinders in staggered arrangements," *J. Fluid Mech.*, vol. 765, pp. 114-149, 2015.

[6] A. V. de Paula, L. A. M. Endres, and S. V. Möller, "Experimental study of the bistability in the wake behind three cylinders in triangular arrangement," *J. Braz. Soc. Mech. Sci. Eng.*, vol. 35, no. 2, pp. 163-176, 2013.

[7] G. Pouryoussefi, M. M, and A. M.A, "Experimental Investigation of Force Coefficients for Groups of Three and Four Circular Cylinders Subjected to a Cross-flow," *Mech. Aerosp. Eng. J. Fluids Aerodyn.*, vol. 5, no. 1, pp. 87-95, 2009.

[8] K. Lam, W. Q. Gong, and R. M. C. So, “Numerical simulation of cross-flow around four cylinders in an in-line square configuration,” *J. Fluids Struct.*, vol. 24, no. 1, pp. 34-57, 2008.
[9] K. Lam and L. Zou, “Experimental study and large eddy simulation for the turbulent flow around four cylinders in an in-line square configuration,” *Int. J. Heat Fluid Flow*, vol. 30, no. 2, pp. 276-285, 2009.
[10] L. Zou, Y. Lin, and H. Lu, “Flow Patterns and Force Characteristics of Laminar Flow Past Four Cylinders in Diamond Arrangement,” *J. Hydrodyn.*, vol. 23, no. 1, pp. 55-64, 2011.
[11] Y. Kahil, S. Benhamadouche, A. S. Berrouk, and I. Afgan, “Simulation of subcritical-Reynolds-number flow around four cylinders in square arrangement configuration using LES,” *Eur. J. Mech. - BFluids*, vol. 74, pp. 111-122, 2019.
[12] C. Geuzaine and J. F. Remacle, “Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities,” *Int. J. for Numerical Methods in Engineering* 79(11), 1309-1331, 2009.
[13] F. Archambeau, N. Méchitoua and M. Sakiz. “Code Saturne: A Finite Volume Code for the computation of turbulent incompressible flows - Industrial Applications,” *Int. J. on Finite Volumes*, 1 (1), 2004.
[14] J. Ahrens, B. Geveci and C. Law. *ParaView: An End-User Tool for Large Data Visualization*. In Visualization Handbook. Elsevier, 2005.