

Numerical Study of Turbulent Flow in Open Channels with Limited Vegetation near The Banks

Yacine Kahil^{1,2}, Abdelkrim Benlefki^{1,2}, Djilali Zerrouki³, Ikram Sir¹, Mohamed Hamel², Tayeb Yahiaoui³

¹Department of Civil, Mechanical and Transport Engineering, Tissemsilt University - Ahmed Ben Yahia ElWancharissi
Tissemsilt 38000, Algeria

kahil.yacine@univ-tissemsilt.dz; benlefki.abdelkrim@univ-tissemsilt.dz; djilali.zerrouki@gmail.com

²Applied Mechanics Laboratory, University of Science and Technology - Mohamed Boudiaf
Oran 31000, Algeria

³Laboratory of Maritime Sciences and Engineering, University of Science and Technology - Mohamed Boudiaf
Oran 31000, Algeria

ikramsir38@gmail.com; hamel_moh78@yahoo.fr; yahiaoui_tayeb@yahoo.fr

Abstract - In this work, RANS approach was adopted to study turbulent flow around cylinders arranged in a square in-line configuration, in open channels with limited vegetation near the banks. The case study concerns an 8x8 cylinders, with a spacing ratio of $P/D = 1.5$ and a Reynolds number of 650. The analysis focused primarily on the mean velocity, pressure, and turbulent kinetic energy fields, as well as the longitudinal and transverse velocity components. In addition, the spatial evolution of the flow was examined to better understand fluid-structure interactions and the effects induced by vegetation on the banks. The simulation results show a good agreement with experimental outcomes from a physical model, specifically for the mean flow terms. It has been shown that the RANS/ $k-\omega$ SST modelling approach provides a satisfactory level of predictive accuracy for engineering applications.

Keywords: CFD, Code_Saturne, turbulent flow, vegetation patches, bank protection

1. Introduction

Flow around cylinders, whether arranged in in-line or staggered, is an important topic for in-depth numerical and experimental research and one of the most studied problems in fluid mechanics as stated by Adobes et al. [1], Kahil [2]. As application in hydraulics and environmental engineering; the study of turbulent flow in open channels with limited vegetation patches near a bank. A brief bibliography was provided covering numerous works in this field. For the experimental studies,

Arora and Kumar [3] explored the effect of emergent vegetation on bank erosion using sediment extraction, examining how static vegetation affects water flow and the morphological changes resulting from erosion. Sharp gradients in current velocity and turbulence factors were observed at the boundaries between plant-covered and non-plant-covered areas. The analysis showed that vegetation increases the velocity of the main channel. A sudden change in shear stress causes instability and erosion of the embankment at the bank level. Although vegetation helped reduce river bank erosion, erosion in the main channel initially increased by approximately 67% due to vegetation. Erosion also increased by 7.66% in the center of the main channel as it proved effective in maintaining the shape of the banks.

Zhang et al. [4] conducted an experimental study of the combined effect of vegetation density and elongation with the spacing ratio on the flow behavior and structure after impact of a rectangular piece with a central channel. It was found that the spacing ratio plays a crucial role, which cannot be induced at a low solid volume ratio. Increasing vegetation length improves generation due to the higher velocity gradient in the post-shock region, which increases the internal velocity of the vegetation.

Koken and Constantinescu [5] experimentally investigated the effect of the immersion ratio (D/h) on the flow structure and turbulence around a rectangular array of rigid cylinders next to an open channel. During immersion, a vertical shear layer forms between the upper surface of the matrix and the free surface, which enhances the three-dimensionality of the flow and affects momentum exchange. In the emergent case, only a horizontal shear layer appears, which deflects the flow laterally, as a balance is reached between the flow entering from the top and exiting from the side. Indeed, the effect of horizontal shear vortices within the matrix is reduced compared to the emergent case. As the size and width of the shear

vortices increase at low immersion rates and then rapidly decrease, strong up-and-down motions appear in the matrix, and the rotation of the near-side secondary cells peaks at this ratio. The total drag increases with immersion, but the contribution of the front cylinders gradually decreases.

Safi and Tominaga [6] conducted an experimental study. When using a breakwater for shore protection, it is ideal to achieve a gentle decrease in the main current velocity while minimizing turbulence around the structure. This can be achieved by modifying the barrier's permeability or by appropriately designing it. It has been shown that the type of pile arrangement in the pile group breakwater significantly affects the downstream flow structure. However, these effects have not yet been adequately studied. In this study, the effects of breakwaters with different pile combinations on flow characteristics were investigated through experiments.

Safi and Tominaga [7] conducted an experimental study on the gradual reduction of the main stream velocity up to the river bank. This is an important objective when using breakwaters for river bank protection. This objective can be achieved by making changes to the permeability or design of the barrier. In this study, the effects of different types of breakwaters on flow deceleration were explored through practical experiments. Several aspects were considered, such as pile density, row arrangement and spacing, as well as their effects over the long river course. The results show that the pile arrangement pattern is a critical factor in controlling the volume and flow configuration behind the pile group breakwater. Through these arrangements, a gradual and smooth reduction in the main stream velocity can be achieved.

Safi and Tominaga [8] examined the effects of different types of dams built on the banks of the Kiso River in Japan over the decades to control bank erosion. This study reviews the effects of different dam types on flow and sediment deposition. Different numbers of piles per group, defined as pile density, were compared for two types of pile arrangement. The results indicate that dams can reduce flow velocity and increase sand deposition in the riverbed along the bank.

Safi and Tominaga [9] conducted an experimental study on the impact of layout and density on water flow behavior in rivers, focusing on bank protection. The results indicate that the staggered configuration favorably modifies the downstream velocity distribution, limiting erosion by reducing velocity near the bank while allowing the main flow to continue without too much disturbance. Furthermore, this layout allows for the use of fewer piles to achieve the same effect, which reduces resistance to the main flow and minimizes the impact on the natural flow. In summary, the staggered arrangement proves more effective for bank protection, turbulence reduction, and streambed stability, while preserving the natural flow.

Cassan et al. [10] conducted field measurements over an 8-month period using an acoustic accelerometer. The analysis uses Reynolds stresses to determine shear velocities and vegetation heights, to calculate discharge, and to study the effect of vegetation height on water flow.

For numerical studies Liu et al. [11] conducted an artificially planted patches of limited vegetation near the shoreline. This method is widely used as semi-natural bank protection measures, which provides insight into their impact on local flow structures. The results indicate that increasing vegetation density leads to reduced flow velocity and increased dispersion in intermediate and turbulent fields. Stem arrangement patterns also affect how water flow interacts with these plant groups.

Gubashi et al. [12] developed a two-dimensional numerical model to simulate open channel flow, incorporating the combined effect of vegetation represented by vertical cylinder. The presence of vegetation significantly influences the distribution of velocity, turbulent kinetic energy, and dissipation, which impacts the flow dynamics in the channel.

Wang et al. [13] analyzed the effect of vegetation on particle transport in curved U-shaped channels using CFD. The results show that the vegetation reduces the intensity of secondary flow, which affects the particle distribution. It also limits vertical sediment transport despite increasing turbulent kinetic energy, which leads to sediment accumulation in the lower layers.

Rahimi et al. [14] performed numerical modeling of open-channel flows with vegetation. In this study, Ansys Fluent software with a k- ϵ model was used to simulate different vegetation configurations, with the aim of analyzing the complexity of flows in vegetated areas. The numerical results showed good agreement with the expected values of turbulent kinetic energy, based on analytical and experimental studies.

Gillani et al. [15] conducted a numerical study for flow control in an open rectangular channel using two impermeable dikes on the displacement of the recirculation zone. The Reynolds turbulent stress model was used using 3D FLUENT software (ANSYS). To analyze the stream flow, mean flow velocity profiles were plotted at selected locations. The results showed that the velocity profiles were affected by changes in the shape and arrangement of the cylindrical zone.

Cui et al. [16] analyzed vegetated water fluxes using different hydrodynamic parameters. The results showed that the mixed layer predominates in vegetation. Although the model successfully described the velocity distribution, some data dispersion was observed.

Yia et al. [17] conducted a numerical study of the hydrodynamics influenced by a bed deformed by patches of vegetation. The water dynamics on deformed layers with a vegetation zone close to the bank are significantly different from those on flat layers, where the topography of the deformed layer results in a clear decrease in longitudinal velocity and shear stress in the open zone layer.

Nadim and Zavid [18] numerically studied an open-channel flow with discontinuous vegetation to estimate the effects of vegetation condition, velocity structure, and turbulence characteristics on a vegetated open-channel flow. The study was conducted using FLUENT software to determine the optimal distribution of vegetation within the flow to reduce flood risk. A three-dimensional Reynolds turbulence stress model was used to analyze water flow through vegetation patches, with a double vertical arrangement on both sides of the channel, while testing two different patch distribution models (in-line and staggered). The results showed lower flow velocities in the gap regions due to the protective effect provided by the vegetation patches. Flow velocity in the unvegetated main channel also increased.

Tariq et al. [19] performed 3D numerical modeling of flow characteristics in an open channel with vegetation patches. These patches influence flow velocity and discharge capacity, thereby helping to reduce flood risks. In this study, ANSYS FLUENT software was used to simulate turbulent water flow around circular vegetation patches with progressively increasing densities. The Reynolds stresses and turbulent kinetic energy in the breach zone decreased, providing a stable environment that favors sediment deposition and aquatic plant growth.

Li et al. [20] conducted a numerical study of the effects of vegetation patch configurations using a 2D numerical model. The model was validated through laboratory experiments and then used to study the effect of two vegetation patterns, one where plants are evenly distributed throughout the plot and the other where plants grow only at the edges. The study included 14 scenarios, and the results showed that the striped pattern leads to channel widening, a steeper slope, and a decrease in water depth with increased sediment transport rates.

Anjum and Tanaka [21] conducted a numerical study of the river vegetation. This vegetation influences adaptation to channel flow and contributes to water management and river restoration. 3D FLUENT and the Reynolds stress turbulence model were used to study water flow through discontinuous vegetation occupying half the channel width. The study included three types of vegetation while maintaining the same density. The results showed that the water flow was more complex. In the spaces between the vegetation patches, flow velocity and turbulent kinetic energy decreased, suggesting a positive role in supporting aquatic life and sediment deposition.

Ghani et al. [22] conducted a numerical study of flow around vegetation patches. The numerical model was validated by comparing it with experimental data at different plant densities and flow velocities, and the results showed good agreement with the experiments. The simulation revealed a decrease in the average flow velocity behind the vegetation patches. The Reynolds stresses were found to be negative at the top of the submerged vegetation, reaching their maximum values at the spacing ratio.

Anjum and Tanaka [23] analyzed the effect of vegetation on flow turbulence in an open channel, using a k-epsilon (ϵ) model developed in ANSYS FLUENT 3D software. After model validation, the effect of vegetation density and vegetation patch length on flow turbulence was investigated. The results showed that flow velocity changes significantly above and below vegetation, with strong fluctuations at the top of submerged features. The decrease in flow velocity was also more pronounced in the presence of tall plants than short plants, and the velocity was higher in vegetated patches than in empty areas. The flow structure in large patches was also more stable than in small ones, while the variation in turbulence caused by the change in distribution shape was small.

Kasiteropoulou et al. [24] conducted a numerical study of turbulent flow in an open channel with a vegetated bottom, modeled with small-diameter vertical cylinders. The 3D simulation was performed using ANSYS-CFX software, based on the finite volume method. The relationship between the hydrodynamic model and pollutant transport and sedimentation processes in aquatic environments was also analyzed.

Previous research on the study of turbulent flows in open channels has shown that the interaction between flow and vegetation leads to complex changes in hydrodynamics, which has led to the use of numerical models such as CFD. The present paper deals with the study of turbulent flows in open channels with limited vegetation near the banks (Fig. 1). The open channels flow model consists of solving the three-dimensional flow equations with appropriate boundary conditions. The vegetation is modeled as 8x8 in-line cylinders. In current CFD analysis, open-source CFD tools are used. The Gmsh tool is used to build the geometry and to generate the computational grid, while the computational tests are performed with Code_Saturne software and the results are analyzed by ParView. The numerical results are compared with experimental data in literature with the aim of indicating the capability of the code and the modeling approach to capture the details of the flow field under such a configuration.



Fig. 1: Group of piles along the Kiso River in Japan. [6]

2. Case study

In this section, a numerical model used to simulate turbulent flow in open channels is validated. The RANS approach is used to simulate the behavior of turbulent flow. The validation process aims to compare the results obtained from benchmark studies, which investigated the influence of geometric factors on flow under similar conditions. This comparison will allow evaluating the accuracy of the model and its ability to represent physical phenomena, thus improving the reliability of the results.

2.1. Geometric parameters

In figure 2a a 3D view of the computational domain carried out in the work of Liu et al. [11]. Regarding our study, figure 2b shows the configuration of (8x8) cylinders in a square arrangement with a spacing of $P/D = 1.5$ (spacing ratio measured between the center of the cylinders and the diameter). The dimensions of the computational domain are $(225D * 48D * 1D)$ in (X), (Y) and (Z) respectively. The upstream length of the vegetation zone is $75D$ and $150D$ downstream. A uniform velocity is imposed at the inlet along the (X) direction and a wall condition is applied on the sides of the channel along (Y) and for the (Z) direction the periodicity has been applied. The flow is modeled under turbulent conditions for an incompressible fluid without heat transfer. The analysis is carried out for a Reynolds number $Re = 650$.

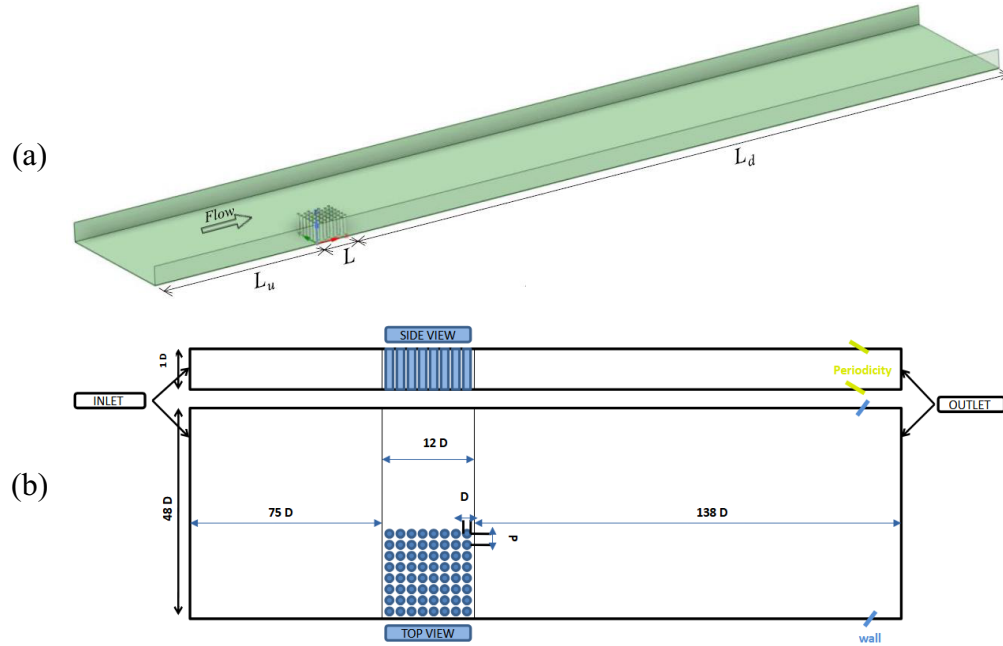


Fig. 2: Computation domain of the case study (8x8 in line cylinders).
(a) 3D view of the domain of Liu et al. [11]. (b) 2D view of the present study.

In this study, the objective is to evaluate the performance of different pile breakers to control the flow around the cylinders. These pile groups were studied by being connected to a side wall, as shown in figure 3.

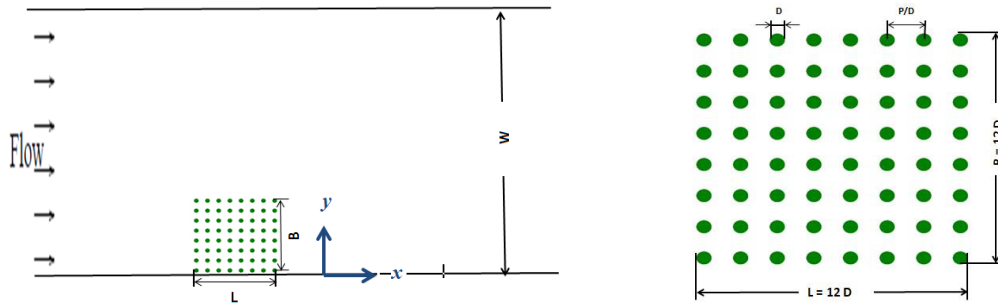


Fig. 3: Zoom on the computation domain of the case study (8x8 cylinders in line).

2.2. Mesh generation

Figure 4 shows a zoomed view of the vegetation zone, on the XY plane. Different meshes are created (coarse, medium and fine) in order to prepare the mesh sensitivity study. This study allows the optimal distribution of cells to be chosen in order to obtain the best results.

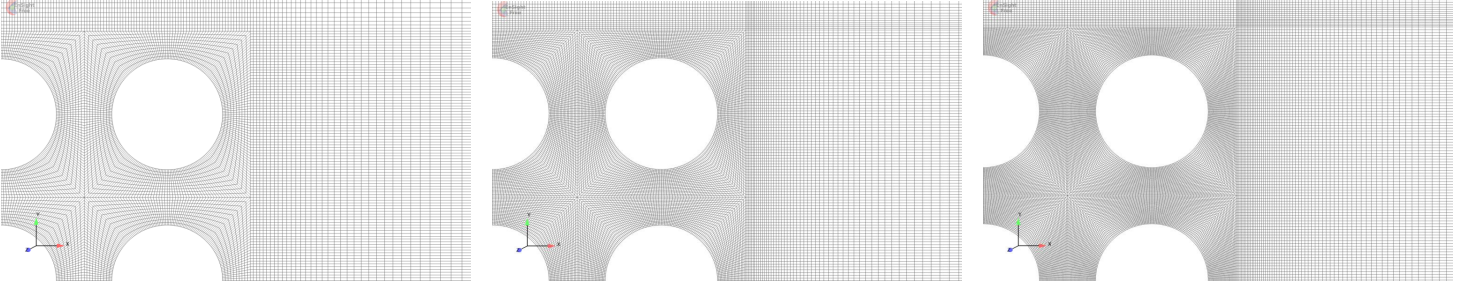


Fig. 4: Zoomed mesh on the XY plane. (a) coarse, (b) medium, (c) fine.

Table 1 shows the number of cells for the three meshes (coarse, medium and fine) on the XY plane.

Table 1: Number of cells in each direction for different meshes

Type of mesh	N cells (Nx * Ny)
Coarse	315200
Medium	522000
Fine	748800

2.3. 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.

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:

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

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

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.

2.4. Sensitivity study

Determining the appropriate mesh fineness is a crucial but complex step. It is advisable to choose a mesh fine enough to ensure the accuracy of the results and correctly represent all the physical phenomena, while avoiding excessive computation times. Mesh optimization is crucial to identify the most suitable model. In practice, the mesh is generally refined near the walls, as well as in the recirculation zones, in order to effectively simulate the flow in the boundary layers and to capture areas with strong gradients. The mesh density can thus vary between coarse, medium and fine according to the needs. However, the grid refinement should not cause significant variation in the results obtained.

As part of the mesh sensitivity study, a comparison was made between the three mesh levels: coarse, medium, fine. The objective was to determine the most suitable mesh for subsequent calculations. The longitudinal mean velocity profiles along the wake axis (at $y = B/2$) corresponding to this analysis are presented in Figure 4. This figure shows that an increase in mesh refinement leads to a slight modification of the velocity profile.

According to the longitudinal mean velocity profiles along the centerline shown in Figure 4, a variation is noticed in the wake of the vegetation zone. Since the profiles (medium and fine) are almost identical, the medium mesh is chosen for further processing and comparison of the results.

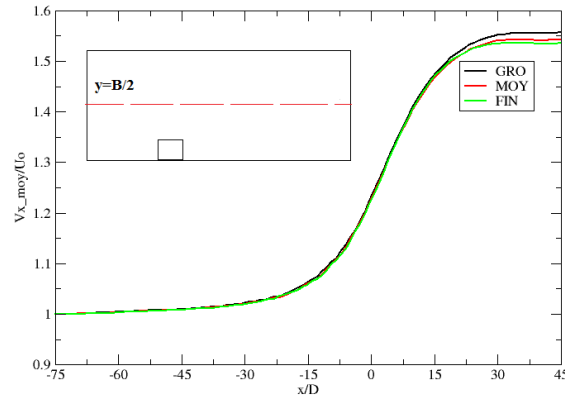


Fig. 4: Profiles of the mean longitudinal velocity along the centerline.

3. Results and discussion

Figure 5 represents the distribution of the mean longitudinal velocity and compared to the results of Safie and Tominaga [6]. The simulation results are presented in the cross section $x/D=1.2B$. A decrease in the velocity in the left part of the figure is noticed, indicating the influence of plants near the bank. Then, the velocity shows a notable increase to reach a maximum value of about 1.5 of the reference velocity U on the banks. A progressive decrease in the velocity is due to the flow approaching the other bank. A similarity between the simulation results and the experimental results for the velocity is noted, which testifies to the ability of the numerical model used to simulate the effect of vegetation cover.

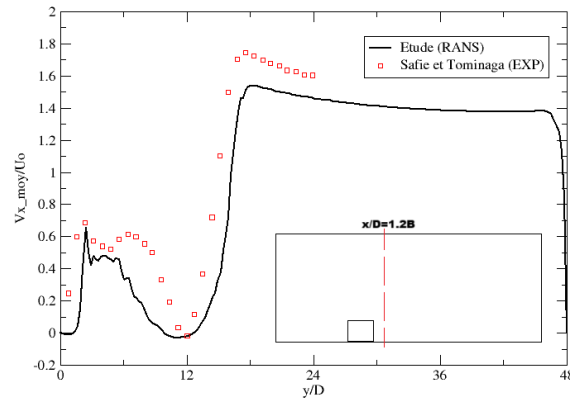


Fig. 5: Distributions of the mean longitudinal velocity at $x/D=1.2B$. (— present study, $\square$ Safie and Tominaga [6])

Figure 6 illustrates results of the numerical simulation using the Reynolds-Averaged Navier-Stokes (RANS) model, showing the flow field for a study with 64 cylinders arranged in a square, inline configuration. The analysis of the mean flow quantities identifies a number of important hydrodynamic features. The mean longitudinal velocity field (Vx_moy) demonstrates significant acceleration in the interstitial wakes immediately downstream of the cylinder arrays, indicating

flow constriction. However, the expansive zones with low velocity are in direct correspondence with the cylinder domains, indicating that the momentum of the flow has been absorbed by the vegetative elements. The mean lateral velocity field (V_{y_moy}) demonstrates mostly weak velocity magnitude throughout most of the domain. There are however small patches of stronger vertical motion directly above and in the near wake of the array, indicating coherent turbulent structures and vertical mixing caused by the obstacles. The mean pressure field (P_moy) exhibits high pressure upstream and on the front surfaces of the cylinders and there is a significant drop in pressure in the lee of the array where a low pressure wake zone exists due to flow separation and vortex shedding. The instantaneous pressure distribution, which demonstrates substantial spatial and temporal variations caused by turbulent eddies. This is quantitatively illustrated in the turbulent kinetic energy (TKE) field which establishes regions of high magnitude turbulence directly downstream of the cylinders, where shear layers interact and breakdown. The magnitude distribution shown for TKE, in particularly strong plume propagating downstream from the vegetated bank zone, is significant, which is supported by Liu et al. [11].

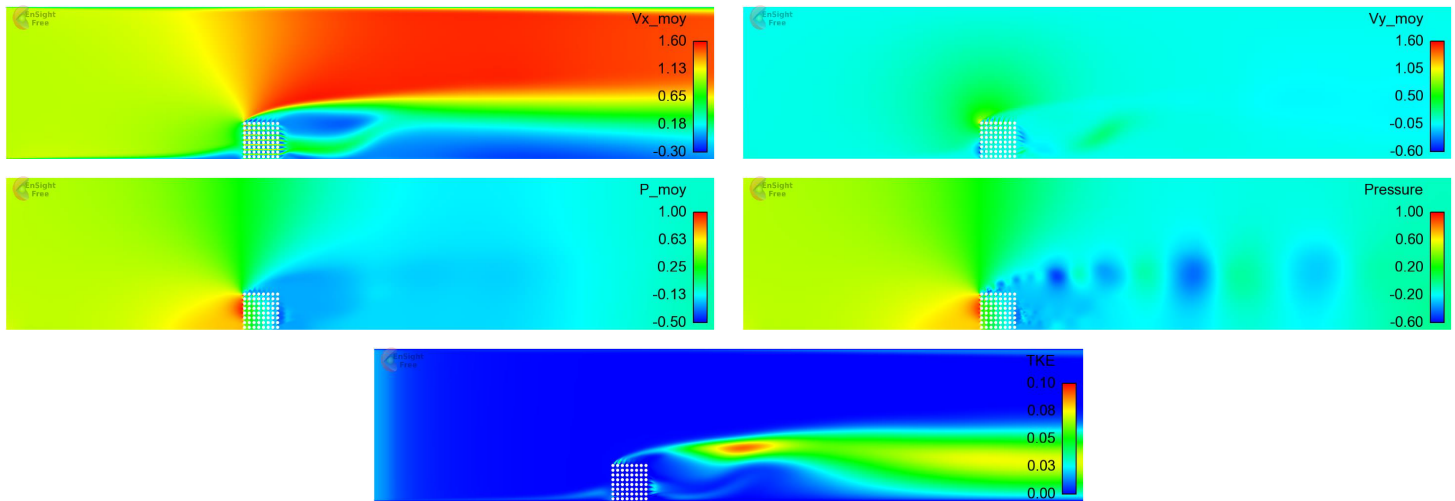


Fig. 6: Different fields for the case of 8x8 cylinders in a square in-line arrangement.

4. Conclusion

In this work, we used numerical simulation to study turbulent flow around cylinders arranged in a square in-line configuration, in open channels with limited vegetation near the banks. This study aims to understand the flow behavior and explain the physical phenomena using open source/free software.

The case study concerns an 8x8 cylinders, with a spacing ratio of $P/D = 1.5$ and a Reynolds number of 650. The RANS approach was adopted in this study, using the turbulence model $k - \omega SST$. The analysis focused primarily on the mean velocity, pressure, and turbulent kinetic energy fields, as well as the longitudinal and transverse velocity components. In addition, the spatial evolution of the flow was examined to better understand fluid-structure interactions and the effects induced by vegetation on the banks.

The post-processing and analysis of data were presented to examine recirculation zones, stagnation points and mean pressure distribution on the cylinders.

The results presented are in good agreement with the experimental data and demonstrate that the RANS approach achieves satisfactory results, thus contributing to a better understanding of flow behavior and the explanation of physical phenomena, with an accuracy relatively close to the results obtained by advanced methods.

References

- [1] A. Adobes, F. Jusserand, S. Benhamadouche, Y. Kahil and S. Belouah, "Computation of Fluid Forces Acting on an Infinite Cylinder Submitted to a Single Phase Cross Flow," *Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting collocated with 8th International Conference on Nanochannels, Microchannels, and Minichannels. ASME 2010 7th International Symposium on Fluid-Structure Interactions, Flow-Sound Interactions, and Flow-Induced Vibration and Noise*, Montreal, Quebec, Canada, 2010, vol. 3, Parts A and B, pp. 241-250.
- [2] Y. Kahil, "Simulation des grandes échelles d'écoulements turbulents autour de cylindres circulaires à un nombre de Reynolds sous critique," Ph.D. dissertation, Institut d'Alembert, Université Pierre et Marie Curie, Paris, France, 2011.
- [3] S. Arora and B. Kumar, "Effect of emergent vegetation on riverbank erosion with sediment mining," *Sci Rep.*, vol. 14, 11193, 2024.
- [4] Y. H. Zhang, H. F. Duan, X. F. Yan and A. Stocchino, "Experimental study on the combined effects of patch density and elongation on wake structure behind a rectangular porous patch," *Journal of Fluid Mechanics*, vol. 959, no. A36, 2023.
- [5] M. Koken and G. Constantinescu, "Influence of submergence ratio on flow and drag forces generated by a long rectangular array of rigid cylinders at the sidewall of an open channel," *Journal of Fluid Mechanics*, vol. 966, no. A5, 2023.
- [6] O. Safie and A. Tominaga, "Effects of Pile Arrangement on The Flow Around a Pile –Group Groyne," *Journal of JSCE*, vol. 8, no. 1, pp. 207-220, 2020.
- [7] O. Safie and A. Tominaga, "Flow control by pile-group used for riverbank instability management," *IOP Conf. Ser.: Earth Environ. Sci.* 2020, vol. 511, 012002.
- [8] O. Safie and A. Tominaga, "Effects of Pile Density and Arrangement on Flow Characteristics and Sediment Deposition Around a Pile –Group Dike," *Journal of Japan Society of Civil Engineers, Ser. A2 (Applied Mechanics (AM))*, vol. 75, no. 2, pp. I_487-I_498, 2019.
- [9] O. Safie and A. Tominaga, "Effect of Pile Arrangement on Flow Characteristics Around Pile –Group Dike," *Journal of Japan Society of Civil Engineers, Ser. A2 (Applied Mechanics (AM))*, vol. 74, no.2, pp. I_439-I_448, 2018.
- [10] L. Cassan, G. Belaud, J-P. Baume, C. Dejean, F. Moulin, "Velocity profiles in a real vegetated channel," *Environmental Fluid Mechanics*, vol. 15, no. 6, pp. 1263-1279, 2015.
- [11] M. Liu, W. Huai, H. Tang, Y. Wang and S. Yuan, "Numerical Study of Mean and Turbulent Flow Adjustments in Open Channels with Limited Near-Bank Vegetation Patches," *Physics of Fluids*, vol. 36, no. 095117, 2024.
- [12] K. R. Gubashi, S. A. Al-Hashimi, S. Mulahasan and A. T. Al-Madhhachi, "A Numerical Model To Evaluate Flow Characteristics In A Circular Vegetation Patch For Flume Experiments," *Journal of Applied Science and Engineering*, vol. 28, no. 1, pp. 75-91, 2024.
- [13] M. Wang, Q. Yu, Y. Xu, N. Li, J. Wang, B. Cao, L. Wang and E.J. Avital, "A Numerical Study on the Influence of Riparian Vegetation Patch on the Transportation of Suspended Sediment in a U-Bend Channel Flow," *Fluids*, vol. 9, no. 5, pp.109, 2024.
- [14] H. Rahimi, C.M.S. Fael, C.S.B. Taborda, S. Yuan, X. Tang, P.K. Singh, E. Fardoost and C.A.V. Santos, "Numerical Modelling of Turbulence Kinetic Energy in Open Channel Flows with Mixed-Layer Vegetation," *Water*, vol.15, no. 14, pp. 2544, 2023.
- [15] H. S. M. Gillani, Z. ul-Hassan, H. R. A. Sarwar, M. S. Jameel, W. Hasan, E. Manzoor and I. Khan, "Flow Control in a Rectangular Open Channel using Two Impermeable Spur Dikes: A Numerical Study," in *Proceedings of the Pakistan Academy of Sciences: A Physical and Computational Sciences*, 2023, vol. 60, no. 2, pp. 17-25.
- [16] H. Cui, S. Felder and M. Kramer, "Predicting flow resistance in open-channel flows With submerged vegetation, *Environmental Fluid Mechanics*," *Environmental Fluid Mechanics*, vol. 23, no. 1, pp. 757–778, 2023.
- [17] Z. Yia, Y. Sun, X. Wang, D. Liu and X. Yan, "Numerical analysis of hydrodynamics influenced by a deformed bed due to a near-bank vegetation patch," *Water Supply*, vol. 22, no. 2, pp. 1546-1556, 2022.

- [18] N. Ahmed and Z. I. Bangalee, "Numerical Simulation of Discontinuously Vegetated Open Channel Flow to Estimate Effects of Vegetation Condition on Flood Mitigation," *Journal of Hydraulic Engineering*, vol. 151, no. 5, pp. 1-14, 2022.
- [19] H. Tariq, U. Ghani, N. Anjum and G. A. Pasha, "3D numerical modeling of flow characteristics in an open channel having in-line circular vegetation patches with varying density under submerged and emergent flow conditions," *J. Hydrol. Hydromech.*, vol. 70, no. 1, pp. 128-144, 2022.
- [20] J. Li, N. Claude, P. Tassi, F. Cordier, A. Vargas-Luna, A. Crosato and S. Rodrigues, "Effects of Vegetation Patch Patterns on Channel Morphology: A Numerical Study," *Journal of Geophysical Research: Earth Surface*, vol. 127, no. 5, pp. e2021JF006529, 2022.
- [21] N. Anjum and N. Tanaka, "Investigating the turbulent flow behaviour through partially distributed discontinuous rigid vegetation in an open channel," *Journal of Hydrodynamics*, vol. 36, no. 8, pp. 1701-1716, 2020.
- [22] U. Ghani, N. Anjum, G. A. Pasha and M. Ahmad, "Investigating the turbulent flow characteristics in an open channel with staggered vegetation patches," *River Res Applic.*, vol. 35, no. 7, pp. 966-978, 2019.
- [23] N. Anjum and N. Tanaka, "Study on the flow structure around discontinued vertically layered vegetation in an open channel," *J Hydrodyn.*, vol. 32, no. 3, pp. 454-467, 2019.
- [24] D. Kasiteropoulou, A. Liakopoulos, N. Michalolias and E. Keramaris, "Numerical Modelling and Analysis of Turbulent Flow in an Open Channel with Submerged Vegetation," *Environ. Process.*, vol. 4, no. 1(suppl.), pp. 47-61, 2017.