Computational Fluid Dynamics Modeling of Submerged Vegetation in Open-Channels

Izet Mehmetaj[1], Mirjam Ndini[1]

[1]*Department of Civil Engineering, EPOKA University, Albania*

ABSTRACT

This study examines how the submerged vegetation affects the hydraulic properties of the open-channel flow.

To predict the impacts of vegetation presence, a Computational Fluid-Dynamic (CFD) model has been developed. ANSYS CFX 18.2 is used to analyze vertical velocity distribution and other flow characteristics as resistance to varying densities, length and positioning of solid cylindrical rods representing vegetation.

The main objective is to create a CFD approach to identify, determine and measure the parameters that contribute to additional resistance due to vegetation. The aim is to better understand hydraulics in stream flow with vegetation systems and to suggest a design tool for evaluating the potential effect of vegetation on stream water management and ecology in Albania. In this study, it is found that the model for the three considered cases gave different results for each of the evaluated parameters, so a correlation exists between the development of the flow hydraulic properties and the vegetation configurations. Also, the obtained results showed that vegetation has a great influence on the characteristics of the flow.

Keywords: *CFD, flow resistance, open-channel, submerged vegetation, simulation*

INTRODUCTION

Anticipating the pattern of flow through the presence of vegetation is becoming important to hydraulic engineers. That is because vegetation is responsible for additional resistance to the flow in rivers, slows down the water flow, it alters the channel flow and results in patterns that are unlike those in non-vegetated systems.

Moreover, vegetation in open channels induces higher bed roughness, lower flow velocity and has an impact on flood capacity. Thus, a better understanding of hydraulic characteristics in vegetated flow is significant for the management of rivers and their ecosystems.

Albania is a country that experiences flooding very often. The need to have a solution for overcoming the risk of flooding is sometimes very costly and has a huge impact on the environment, so relying on natural processes is more sustainable. Vegetation in riverbanks, emergent or submerged, improves the riverbank stability, reduces erosion, and impacts the floodwater storage. The engineering approach to this issue requires determination of the hydraulic resistance resulting from the existence of vegetation.

This study discusses the presence of submerged vegetation by means of a CFD (Computational Fluid Dynamics) model to predict the additional resistance based on vegetation geometric properties like spacing and height.

The main idea is to establish a CFD modeling approach to identify and quantify the parameters that contribute to the induced resistance due to vegetation presence. A CFD 3D model will be considered such as RANS (Reynolds-averaged-Navier-Stokes) employing a standard k-ε turbulence model for the vegetation simulation on ANSYS CFX software 18.2.

Research shows that the presence of vegetation in the flow increases the hydraulic resistance by causing dragging and turbulence. The mixing induced by the turbulence will have significant effect on ecology, sediment transport in water, pollutants in water, etc. [1].

Different research has been recently carried out on this topic. Authors mainly reviewed aquatic vegetation effect on flow velocity vertical profiles.

For example, Klopstra et al. [2] and Meijer and Van Velzen [3] worked on a method to analytically define the velocity profile into two layers flow domain, one within the vegetation and another above the vegetation. The model is a two-layer model type and consisted on solving the momentum equation in the vegetation layer while considering a logarithmic profile of velocity in the upper layer. Model simulations were consistent with the results of flume experiments reported in these studies.

Lopez and Garcia [4] proposed a model to calculate the mean velocity profiles and turbulence characteristics in open-channel vegetated flows under a 1D framework. This model is a two-equation turbulence approach and was built since the drag due to vegetation is appearing not only in the momentum equation, but also in the equations of a modified k-ε turbulence model. Numerical results including mean velocities, Reynolds stresses, turbulence intensities and different terms in the turbulent kinetic energy, were compared with the experimental observations.

Wilson et al. [5] conducted a study using 3D Finite Volume (FV) numerical model to describe the hydraulic effect of vegetation on the velocity distribution concentrated on the drag-force approach; the model provides solution for the continuity and momentum equations for each cell and for the modelling of turbulence uses the k-ε model.

In recent years, the impact of vegetation in flow resistance and turbulence as well as its influence on sediment transport were inspected.

Yan et al. [6] studied the effects of vegetation rigidity on the hydraulic roughness. The experimental results showed that a rigidity coefficient of vegetation can be used to describe the vegetation rigidity relating to the flowing water.

Nepf [7] discussed the impact of vegetation distribution on sediment movement, by working on methods for estimating bed stress within regions of vegetation.

Izni et al. [8] studied the development of vegetative roughness estimation for hydraulic modelling through laboratory experiments and remote sensing. The study concluded that there is a possibility to utilize the data of remote sensing to accurately estimate the main vegetation properties such as height and density. These would allow to define the parameters to be used as input for the calculations of surface roughness in any practical application.

Timothy et al. [9] employed a three-dimensional model to predict the flow and turbulence dynamics in open-channels. Two different CFD-biomechanical models are suggested on this study. The model's suggestions are based on whether the vegetation is dominated by bending or tensile forces. For bending plants (vegetation is dominated by bending forces), a model structured on the Euler–Bernoulli beam equation has been suggested, while for tensile plants (vegetation is dominated by tensile forces), a N-pendula model has been suggested. The results showed that both models predict the spatial characteristics of the mean flow, turbulence as well as plant-flow interactions.

Panigrahia et al. [9] conducted an experiment in a straight open channel flow using unbending cylindrical rods to investigate the effect of vegetation. In this study, has been investigated the effect of emergent rigid vegetation on the prediction of the effective vegetal drag coefficient for various flow depth combinations.

Richard et al. [10] provided a new method that integrates the complexity of the plant morphologies into a computational fluid dynamics (CFD) model. The complexity of plant morphology, including the vertical and horizontal distribution of their branches and leaves is gathered through earthly laser scanning (TLS) and is kept up in the numerical prediction of flow fields. This approach provides a full flow numerical description of the pressure field. It also allows the vegetative drag force to be measured. The methodology used in this study shows the significance of accurately representing complex plant morphology in hydraulic models.

Evaluating these studies, many physical and numerical models had been proposed and approved. Most of them use numerical modelling within Computation Fluid Dynamics (CFD) framework that enables the representation of realistic vegetation patterns. Also, they validate the application capability of CFD for solving the problem of flow resistance in vegetated systems. But, the majority of these investigations depend on a 2D model development.

NUMERICAL MODELLING

Vegetation is commonly modeled as arrays of rigid, circular cylinders that simplify and generalize plant forms of vegetation in riverine systems. This approach is commonly applied to several laboratory flow conditions, including both, submerged and emergent case [25].

Generally, the model is based on the analysis of forces acting on the volumetric element surrounding a given number of elements of vegetation. Specifically, as water flows through vegetation, drag is produced. The drag creates eddies and velocity gradients that causes the loss of momentum. There exist several numerical approaches that describe the effect of vegetation on flow resistance. Most of recent models are concentrated on a drag-force approach [7].

Wilson et al. [7], introduced a drag-related force term to be implemented on Navier – Stokes equation to describe the flow resistance induced by vegetation. The approach consists of a 3D Finite Volume method. The finite volume method is a numerical method to solve partial differential equations. The finite volume method calculates the values of the conserved averaged variables across the volume. This drag force F_D usually has the form:

$$F_D = \rho \, \frac{u^2}{2} C_D \lambda \tag{1}$$

Where C_D is the drag coefficient and λ represents the projected area of a single plant in stream wise direction. This force can be included as a sub grid force per fluid mass unit in a finite volume (FV) cell.

In this study, the 3D Navier-Stokes equations for steady, incompressible flow in combination with the standard k-ε turbulence model will be solved by the ANSYS CFX code (ANSYS 18.2, 2017).

In this code as in the most typical Computational Fluid Dynamics models, to define the flow field, are solved the Navier-Stokes equations. The Navier-Stokes equations describe the forces acting on a body of water.

In CFD, turbulence models are empirical mathematical models that describe the turbulence in the flow. In these numerical stimulations, there are three approaches for the prediction of the turbulent flow summarized below [28]:

- Reynolds Averaged Navier- Stokes (RANS) simulation
- Large Eddy simulation (LES)
- Direct numerical simulation (DNS)

RANS has been widely used in designs and research since the 70's. K-epsilon (k-ε) turbulence model is the most common RANS model used in Computational Fluid Dynamics (CFD) to simulate turbulent flow conditions.

The standard k-ε model is firstly proposed by Launder and Spalding in 1974 [29]. This turbulence model gives a general description of turbulence by means of two model equations. The first equation considers the kinetic turbulent energy k and the second one for the rate of dissipation of turbulent kinetic energy per unit mass, ε.

The parameters k and ε are determined by solving transport equations:

- For turbulent kinetic energy k:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\ \rho\varepsilon \quad (2)$$

- For dissipation ε:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{K}2\mu_t E_{ij}E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (3)$$

Where u_i represents velocity component, E_{ij} represents component of rate of deformation, μ_t represents eddy viscosity.

$$\mu_t = \ \rho C_\mu \frac{k^2}{\varepsilon} \quad (4)$$

The equations also incorporate some constants σ_k, σ_ε, $C_{1\varepsilon}$, $C_{2\varepsilon}$. The values of these constants have been defined empirically for a variety of turbulent flows by numerous iterations of data [29]. These constants are as follows: $C_\mu = 0.09$; $\sigma_k = 1.00$; $\sigma_\varepsilon = 1.30$; $C_{1\varepsilon} = 1.44$; $C_{2\varepsilon} = 1.92$.

CFD MAIN APPROACH AND METHODOLOGY

In general, most applications of CFD take the same basic approach. The basic procedure is described as follows:

Pre-processing: To define the geometry and physical bounds of the problem computer aided design (CAD) tools can be used. Like that, data can be processed, and the fluid volume (or fluid computational domain) is added. The volume limited by the fluid domain (filled with water, air etc.) is divided and broken up into discrete cells (the mesh generation process). The mesh may be uniform or non-uniform, structured or unstructured. It is made of a combination of hexahedral, tetrahedral, prismatic, pyramidal or polyhedral elements. The modeling is defined physically –the equations describing the fluid motion are set. Boundary conditions are determined either by specifying them on each edge of the computational domain (2-D flow models) or on each face of the domain (3-D flow models). They describe and specify the fluid behavior and properties at all bounding surfaces of the fluid domain. In case of transient problems (time dependent problems), the initial conditions are also defined.

Solving: The simulation is started, and the equations are solved iteratively as a steady-state or transient type of analysis.

Post-processing: As a final step, a postprocessor is used to obtain useful information for the analysis and visualization of the resulting solution.

DEVELOPMENT OF THE MODELS FOR SIMULATION WITH ANSYS CFX (18.2)

In this work, ANSYS CFX (Release 18.2) is used to simulate water flow within the vegetation. The numerical reference is associated with finite volume solution of the three-dimensional Navier-Stokes equations in a Cartesian co-ordinate system, and under a standard k-ε turbulence model.

A rectangular fluid domain with 2.052 m length, 0.232 m width and 0.420 m height with bottom slope 1:5000 is used.

Vegetation is considered as an idealized plant with reference to the approach proposed by Wilson et al. [7]. For that reason, the vegetation is modelled as arrays of rigid, circular cylinders. A rigid vegetation is not changing its position within time.

Following this theoretical viewpoint suggested by Wilson at al. [7], three main cases are modelled:

- Case 1: Submerged vegetation scenario with a parallel simulated pattern
- Case 2: Submerged vegetation scenario with a staggered simulated pattern
- Case 3: Double-layered submerged vegetation with a parallel simulated pattern alternately arranged with tall and short rigid circular cylinders.

These configurations and their sizes are shown in Figure 1. Respectively, there are 838 rods/m^2 for case 2 and 3, and 850 rods/m^2 for case 1.

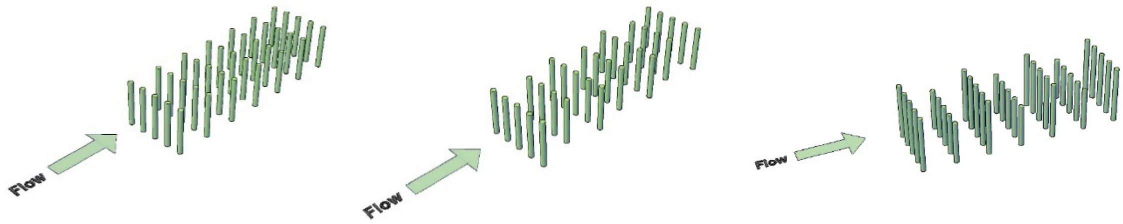

Figure 1. Sketch of vegetation patterns used for modelling (case 1 at the left, case 2 in the middle and case 3 at the right).

Vegetation is simulated with 12 mm diameter rigid circular cylinders of two lengths, k_1= 120 mm and k_2 = 160 mm. In all the three cases the spacing between cylinders is 40 mm in the direction of the stream and 30 mm in the transverse direction of the stream.

In all the scenarios the water is assigned to be used for the fluid domain, at a temperature of 25°C and reference pressure of 1 atm. A normal speed equal to 4 m/s will be imposed as boundary detail for the inlet boundary condition. Also, the no-slip boundary condition is assigned on the both wall type boundaries. Furthermore, the initial conditions are specified globally for the analysis and set equal to 0 m/s for x component, 4 m/s for y component and 0 m/s for the z component. The effect of slope of the channel in the fluid domain will be reflected on resolved gravity components.

Symmetry as a boundary condition is set in the models to specify the zero-shear slip walls in the fluid domain free surface [3]. The flow field will be simulated as transient and over a specified time of 25 s. The time step is set to be 2 s providing 12.5-time steps to be solved.
The time step controls the spacing in time between the variable of interest. The correct choice of the time step will influence the convergence of the simulation.

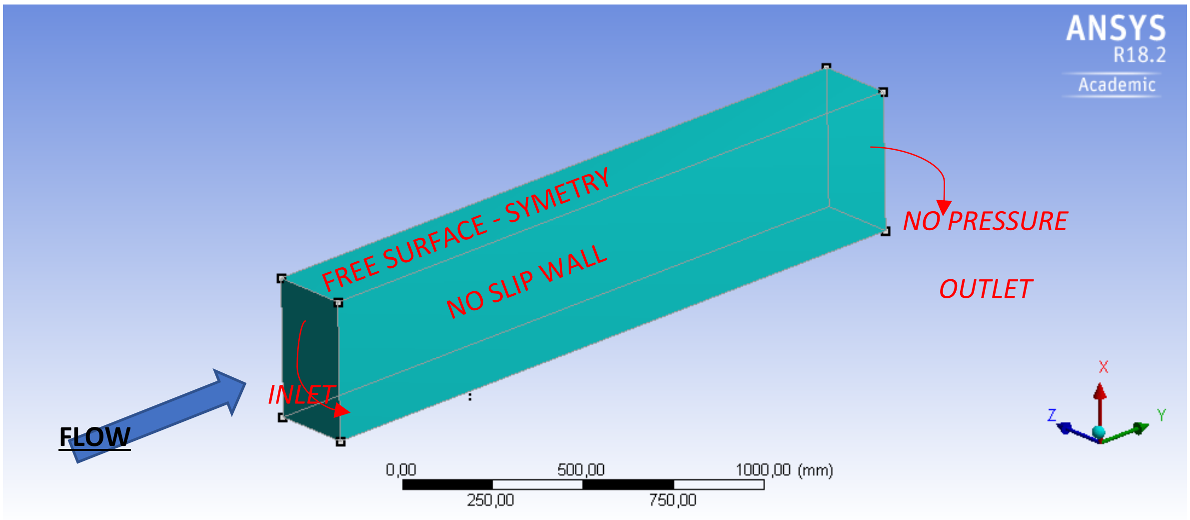

Figure 2. Computational fluid domain schematic

The standard k–ε turbulent model will be used in CFX code, with the model constants $C_\mu = 0.09$, $\sigma_k = 1.00$, $\sigma_\varepsilon = 1.30$, $C_{1\varepsilon} = 1.44$ and $C_{2\varepsilon} = 1.92$. This model is the simplest turbulence model because only initial and boundary conditions needs to be defined.

The three considered cases are employed separately in CFX-Pre. The case 1 refers to time dependent turbulent fluid model with submerged vegetation in a parallel simulated pattern; case 2 refers to time dependent turbulent fluid model with submerged vegetation in a staggered simulated pattern and case 3 refers to time dependent turbulent fluid model with double-layered submerged vegetation in a parallel simulated pattern alternately arranged with tall and short rigid circular cylinders.

In all the cases, water is assigned to be used for the fluid domain, at a temperature of 25°C and reference pressure of 1 atm. For all the three cases, 5 boundary conditions types were inserted for the computational domain: inlet, outlet, wall at the computational domain side faces, the structure wall defined from the vegetation surface region, and the symmetry for the open channel flow free surface.

A normal speed equal to 4 m/s is imposed as boundary detail for the inlet boundary condition. Furthermore, the initial conditions are specified globally for the analysis and domain for Cartesian velocity components and set equal to 0 m/s for x component, 4 m/s for y component and 0 m/s for the z component. Also, the no-slip boundary condition is assigned on both wall type boundaries, that is, the fluid particles on the body move with body velocity. The boundary conditions at the free surface of an open-channel flow suggests that both the pressure and the shear stress are equal to zero everywhere. Here is why a symmetry boundary condition is defined in order to determine a zero value of all the quantities into the symmetry plane. Such definition reassembles the free surface of an open channel flow. The velocity pattern is modelled to be almost uniform along x-axes by adding the free-surface boundary condition. This means that initially, the water will be in movement with a velocity 4 m/s very near to the free-surface and with the depth going increased the velocity will have some curvature providing a 0 value to the fluid domain bottom wall.

RESULTS AND DISCUSSIONS

To study the variations of the velocity distribution at the global scale around rigid vegetation, several analyses of the velocity component data into the open channel fluid domain have been carried out, at XY section's planes along the stream wise direction (Z = 78 mm).

To assign a quantitative value that represent the flow patterns and variations due to vegetation, Manning's roughness coefficient is estimated using The Manning's formula. For all the three considered cases, the estimation is made in the middle of vegetation zone as well as directly after the vegetation.

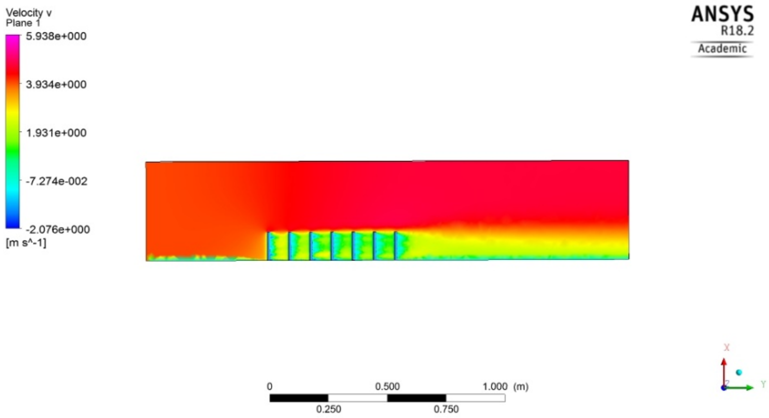

Figure 3. Case 1: Vertical velocity variations at the centrally located XY plane

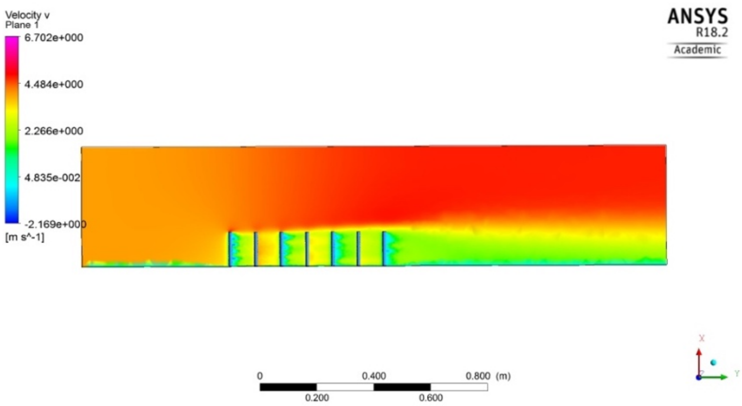

Figure 4. Case 2: Vertical velocity variations at the centrally located XY plane

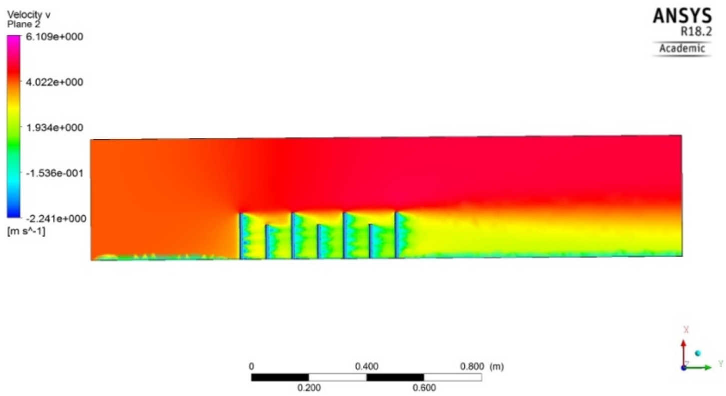

Figure 5. Case 3: Vertical velocity variations at the centrally located XY plane

It is noted that the vertical velocity profile stays almost unchanging in the upper part of the vegetation region. Whereas in the vegetation region the velocity fluctuations are significantly noticeable. A transition occurs at the top of the vegetation height. The fluctuations of the vertical velocity on two distinguished patterns (invariably on the upper part and variable within vegetation) result due to the vegetation existence. At the top of vegetation height is where the transition between these two patterns happens. The presence of vegetation makes the velocity to slow down in comparison with the initial flow velocity. Also, there is a correlation between the velocity development profiles and the vegetation spacing distribution.

In general, it is observed that the flow resistance characterized in terms of n (Manning's coefficient) decreases as the flow depth X is increased. This finding is count by taking as 0 level the bottom of the fluid domain. A similar result is taken at both locations of interest where estimations are made (in the middle of vegetation and directly after the vegetation).

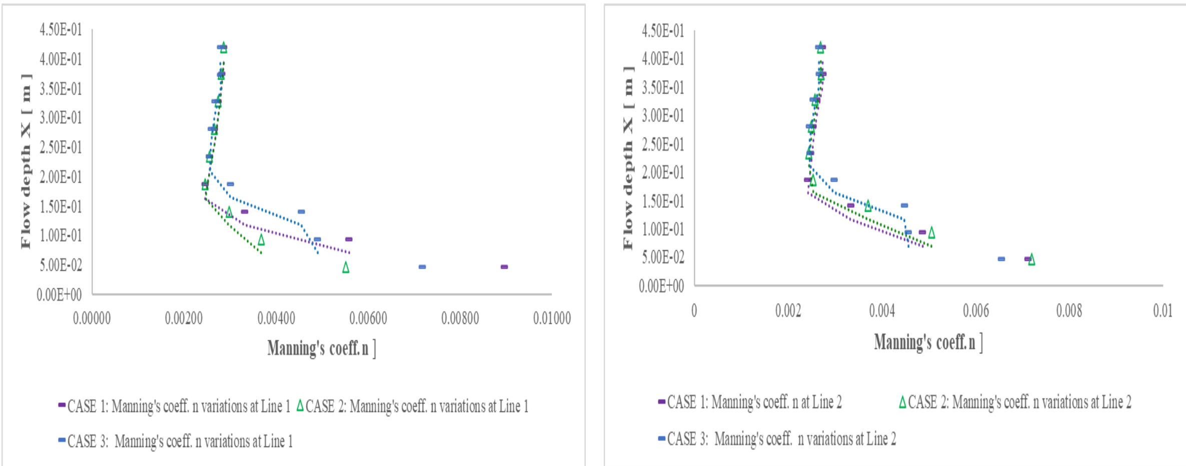

Figure 6. Manning's coefficientn variations within the depth of the open channel domain X, for vegetated zone (left) and non-vegetated zone (right)

A reason is that with increasing flow depth (starting from the bottom of the fluid domain) the absorbing area of vegetation is reduced, that is no vegetation is present as an obstacle. Besides, for the line location directly after the vegetation the effect of flow-rods interaction is different due to rods space arrangement following different flow project areas.

Exceptionally, large values of n are obtained for double-layered vegetation case in the middle of vegetation for the flow depth X range between 0.14-0.187 m. A reason may be the alternation between the tallest and shortest vegetation on this flow depth range that makes the interaction between the rods higher than that in case 1 or 2 as well.

For the case 1, in the middle of vegetation, the values of n are 9 % greater than values of case 2 and 2% less than values of case 3. This observation is notable in the range values of n between 0.003-0.006. However, for the range of n values between 0.002-0.003, the Manning's coefficient leans towards values close to each other.

These variations may occur due to the vegetation properties like height or density. The interaction zone is different between the cases and due to these different vegetation properties, the flow retarding takes different profile to each case. Notable differences are noticed on the vegetation zone. For the given densities, the flow-vegetation produces different values of n.

For the case 2, at location directly after the vegetation, the values of n are 2% greater than values of case 1 and 2% less than values of case 3. Such observation is evident in the range values of n between 0.003-0.007. For the other n range values, they are not varying significantly same as in the zone at the middle of vegetation. For all these readings at the non-vegetated zone, the n values vary and have also different profiles. This proves that vegetation is an important factor while evaluating the flow resistance and modifies it.

To sum up, the obtained results showed that vegetation has a great influence on the characteristics of flow and can be an important feature affecting the flow frictional resistance in channel.

CONCLUSION

In this study, a computational model is provided to predict the impact of submerged vegetation in a constant rectangular cross-section open channel. A 3D-CFD modeling approach is established to quantify resistance and flow patterns in a vegetated open-channel system based on vegetation geometric properties like spacing and height.

For flows through submerged vegetation, model results indicate that vegetation slows down the flow and causes turbulence within the vegetation layer, resulting in reduced velocity gradients. It is found that the model for the three considered cases gave different results for each of the evaluated parameters, so a correlation exists between the development of hydraulic properties and the vegetation spacing distribution. The computational model allows also to predict Manning's coefficient values.

The obtained results demonstrate the application capability of CFD for solving the problem of flow resistance in vegetated systems. It also provides a framework for future study of flow interactions with vegetation to improve the river stability or flood storage capacity. However, further detailed investigations need to be done in river modeling when such solutions must be provided.

To validate the present approach a field study would be of interest. Having the vegetation present in watercourses has advantages and disadvantages. Planting or taking away the vegetation may increase or decrease the flood risk of watercourses in riverine systems. Resistance is a key parameter to define water levels especially in floods considering that the most important effect that vegetation has, is on the flow velocity. To investigate on this, is important to understand and estimate the resistance due to vegetation. By combining the filed data with the proposed CFD approach as a design tool, new insights would come up and at the same time would test the application capability of this tool.

REFERENCES

[1] J. J. Jochen Aberle, "Chapter 21 Hydrodynamics of Vegetated Channels," in Rivers - Physical, Fluvial, Environmental Processes, Springer, 2015, pp. 519-541.

[2] Y. O. R. H.-P. a. O. N. Wilson C.A.M.E., "3D numerical modeling of a willow vegetated river/floodplain system," Journal of Hydrology, pp. 13-21, 2006.

[3] W. W. Y. H. Y. Z. Z. Y. Wenxin Huai, "Analytical model of the mean velocity distribution in an open channel," Advances in Water Resources, Science Direct, pp. 106 - 113, 2014.

[4] B. H. V. N. J. a. V. V. E. Klopstra D., "Analytical model for hydraulic roughness of submerged vegetation, Managing Water: Coping With Scarcity, and," 27th IAHR Congress, San Francisco, USA, 1997.

[5] D. M. E.H. Van Velzen, "Prototype-scale flume experiments on hydraulic roughness of submerged vegetation," 28th IAHR Congress, Graz, Austria, 1996.

[6] F. L. a. M. Garcia, "Mean flow and turbulence structure of open-channel flow through non-emergent vegetation," Journal of Hydraulic Engineering, p. 392–402, 2001.

[7] W. X. Yan Z., "The Effect of Vegetation Rigidity on Flow Resistance," Advances in Water Resources and Hydraulic Engineering. Springer, Berlin, Heidelberg, 2009.

[8] H. M. Nepf, "Hydrodynamics of vegetated channels," Journal of Hydraulic Research, 50:3, pp. 262-279, 2012.

[9] B. Y. M. C. Izni Zahidi, "Vegetative Roughness Estimation for Hydraulic Modelling: A Review," Research in Civil and Environmental Engineering, pp. 1-10, 2014.

[10] R. J. H. S. N. L. &. D. R. P. Timothy I. Marjoribanks, "High-resolution numerical modelling of flow—vegetation interactions," Journal of Hydraulic Research, pp. 775-793, 2014.

[11] K. K.Panigrahia, "Prediction of Velocity Distribution in Straight Channel with Rigid Vegetation," Aquatic Procedia , pp. 819-825, 2015.

[12] R. J. H. J. W. a. T. I. M. Richard J. Boothroyd, "The importance of accurately representing submerged vegetation morphology in the numerical prediction of complex river flow," Earth Surface Processes and Landforms, pp. 567-576, 2016.

[13] T. S. a. P. B. C.A.M.E. Wilson, "Modelling of open channel flow," in Computational Fluid Dynamics: Applications in Environmental Hydraulics, Chichester, England, John Wiley & Sons, Ltd, 2005, pp. 395-428.