3rd Conference on Sustainability in Civil Engineering (CSCE'21)
Department of Civil Engineering
Capital University of Science and Technology, Islamabad Pakistan

NUMERICAL INVESTIGATION OF FLOW BEHAVIOUR IN A ROUGH BANK OPEN CHANNEL WITH VEGETATION PATCHES

[1] *Muhammad Bilal*, [2] Usman Ghani, [3] Muhammad Asif**

1: Civil Engineering Department, University of Engineering and Technology Taxila, hafizbilal39@gmail.com
2: Civil Engineering Department, University of Engineering and Technology Taxila, usman.ghani@uettaxila.edu.pk
3: Housing Urban Development and Public Health Engineering Department, masifshehzad667@gmail.com
* Corresponding author: Email ID: hafizbilal39@gmail.com

Abstract- In natural streams there exists a variation of roughness in stream-wise and transverse directions. In the present study, a numerical technique has been utilized to investigate the flow structure in an open channel comprising of various kinds of hydraulic roughness's. The channel roughness was consisted of roughness elements along the banks of the channel and circular vegetation patches along the center line of the channel. Bank roughness elements of two different lengths l_r were used {0.06 m,0.04 m} having pitch to height ratio p/k {9.67, 10.33}. Results captured by RSM model are presented in the form of depth-averaged velocities, contours of stream-wise velocity distribution and turbulent intensity. The results showed flow velocities in regions downstream of vegetation patches and roughness elements are small and supports the ecological habitat nourishment. Turbulence of minimum magnitude is observed and uniformly distributed in the free unobstructed regions. However, maximum of 5.8% turbulence has been observed in the flow zones of patches and bank roughness elements.

Keywords- Roughness Elements, Vegetation Patches, Computational Fluid Dynamic (CFD), Reynolds Stress Model (RSM), Open Channel Flow.

1 Introduction

Natural streams and rivers benefitted the human beings from the ancient times. To meet the necessities of communities' water from the water bodies are diverted for domestic and industrial use. The human activities over the rivers changed the natural geometry of the conveyance channels which disturbed the equilibrium between erosion and sediment deposition. To protect the water carrying structure a variety of river restoration techniques have been used. These restoration techniques include the protection shape of conveyance channel by a variety of methods i-e pitching, bioengineering, constructing embankments levees and spurs etc. The flow through rivers and fresh-water streams when encountered these different types of river protection structures imparts hydraulic resistance. These elements like vegetation, embayment's, gravel, cobbles etc. behave like a roughness element and encountered for hydraulic roughness and morpho dynamics in the field of hydraulic engineering. For the perspective of basic research, a river has been characterized as a channel which equipped with roughness elements [1]. Bed material of open channel, i.e., gravel bed comprises of small grains act as roughness elements and retard the flow. Variety of roughness elements are being used in the literature which includes rectangular, square, circular, octagonal, spherical, hemispherical in shape [2, 3, 4, 5, 6]. Literature studies suggested that three types of flows are generated by roughness elements depending on their pitch to height ratio P/k, k-type flow, un-reattached flow and d-type flow (P is the pitch between two consecutive elements, k and d are the two types where k is roughness height and d boundary layer thickness) [6].

In river streams commonly there exists a variation of roughness in lateral direction. It may be vegetation on one side or change in the size of gravel beds [7]. In open channels and wetland flows, vegetation also considered as the roughness which imparts resistance to the flow behaviour. Vegetation can be rigid or flexible, which grows naturally on the bed or along the banks of flowing channel. Vegetation in open channel flows changes the entire flow structure, induces turbulence, increases the hydraulic resistance along the wetted perimeter of channel and reduces the conveyance capacity of channel threshold [8, 9]. In open channel flows, vegetation colonializes themselves to behave as mono-specific patches which interacts the flow in very non-linear behaviour. These vegetation patches have a noteworthy effect on marine ecology, they are self-assembled by the clonal growth, resulting in the formation of a vegetation mixture. Few of the previous studies have investigated the flow field and geomorphological changes behind the isolated vegetation patch [10, 11]. Anjum [12] employed numerical CFD code FLUENT to study the mean flow characteristics through a configuration of circular vegetation patch. However, flow structure having more than one patch is considerably different from a single patch.

To the knowledge of the authors, the previous studies considered either vegetation patches or banks roughness elements independently. However, a study of flow characteristics in the presence of both vegetation patches and bank roughness elements has not been made in the past. The present study investigates mean flow behaviour and turbulence characteristics in a rectangular channel having both the vegetation patches and bank roughness elements of different pitch-to-height ratio *P/k*. Computational Fluid Dynamics technique was employed in this study. Reynolds Averaged Navier Stokes (RANS) equations were solved with the help of 3D numerical code FLUENT.

1.1 Numerical Model Validation.

The numerical model was validated against the experimental data of Li [13]. Flow domain comprises of 16 m length, 0.3 m width and 0.4 m height. Rigid circular bamboo cylinders of diameter (d = 4 mm) were used to model the circular vegetation patch (D = 0.06 m) at the center of flow domain. The stem density was described by n, the number of cylinders per bed area and the frontal area per volume a=nd. The bed slope was 0. 001.The vegetation patch occupied the entire water column i-e emergent in all the experimental series. The center of circular vegetation patch was considered as the origin having coordinates (0, 0, 0) in longitudinal, transverse, and vertical directions, respectively. The details of experimental setup could be found in Li [13]. Experiment no. 2 was selected and modelled for validation purpose. The discharge of 18.01 l/s was used and water depth taken was 0.13 m. The large mesh structure and computational time was reduced by selection of only 1 m length of the flow domain. Design-Modeler of ANSYS Workbench was used for creation of flow domain. To mesh the entire flow domain, unstructured mesh with tetrahedral elements was used. Three meshes of different sizes i.e., coarse, medium and fine were generated and consisted of 200 x 60 x 20, 400 x 120 x 40, 600 x 180 x 60 node numbers in stream-wise, lateral and vertical directions respectively. The difference of computational velocities between medium and fine mesh was less than 1%. Based on the mesh independence test, it was assumed that the results of our numerical model are mesh independent. As a result, medium sized mesh having node numbers 400 x 120 x 40 was selected for the simulation purposes.

After the successful generation of mesh, it was imported to the FLUENT for solving the fluid flow phenomena. Different boundary conditions were assigned on different surfaces of the flow domain. The inlet/outlet of the flow domain was mapped and provided with the periodic boundary condition. The component of velocity at the side walls, bed of channel and surfaces of vegetation cylinders is zero, therefore wall boundary condition was applied on these surfaces. The symmetry boundary condition was applied at the top free surface. The Reynolds stress turbulence closure model (RSM) was used for this numerical simulation. The SIMPLE algorithm was selected for the pressure-velocity coupling. For spatial discretization 2nd order upwind and standard wall function was used. All residual plots of numerical computation were set to be converged after attaining the value of 1×10^{-5}. In the lateral direction from origin, vertical lines located at y/D = 0, 0.58D, 1.17D and 1.75D (y is the distance in transverse direction from origin normalized with the vegetation patch diameter D) numerically computed velocities were investigated and compared to the experimentally measured velocities Li [13]. The Figure 1 shows that the normalized numerically computed velocities are in close agreement with the experimentally measured velocities. It indicates that our numerical model can simulate the flow in such situations.

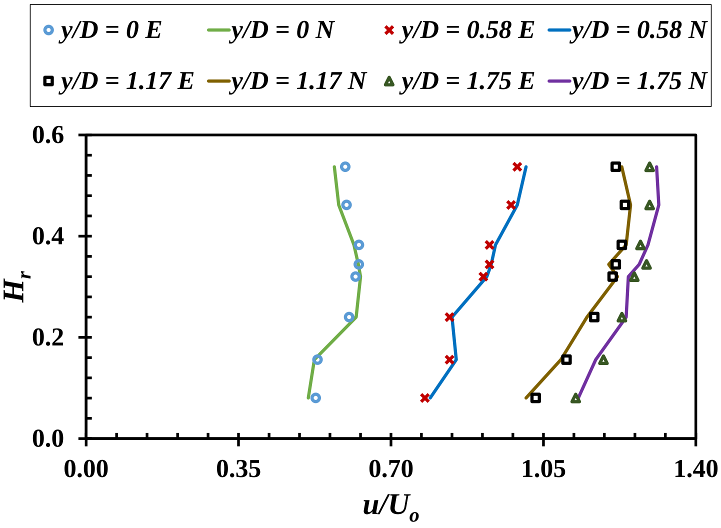


Figure 1: Validation of numerical model by comparing numerically computed results (N) with experimental data(E).

1.2 Numerical Simulation.

After successful validation process, two geometric configurations were considered for simulation purposes. These were comprised of circular vegetation patches along the center line along with bank roughness elements as shown in the Figure 2. The diameter of circular vegetation patch and individual cylinder was 0.06 m and 4 mm respectively. Bank roughness elements were modelled with rectangular elements attached to the side walls as shown in the Figure 2. The height k and width w_r of the roughness elements were kept same in both the cases. Therefor the relative submergence ratio $h/k < 10$ remained same in both the cases. However, roughness elements of two different lengths l_r were used {0.06,0.04} having pitch to height ratio p/k {9.67, 10.33}. The discharge of 18.01 l/s was used in both the cases. The schematic diagram of Case 1 is shown in figure 2.

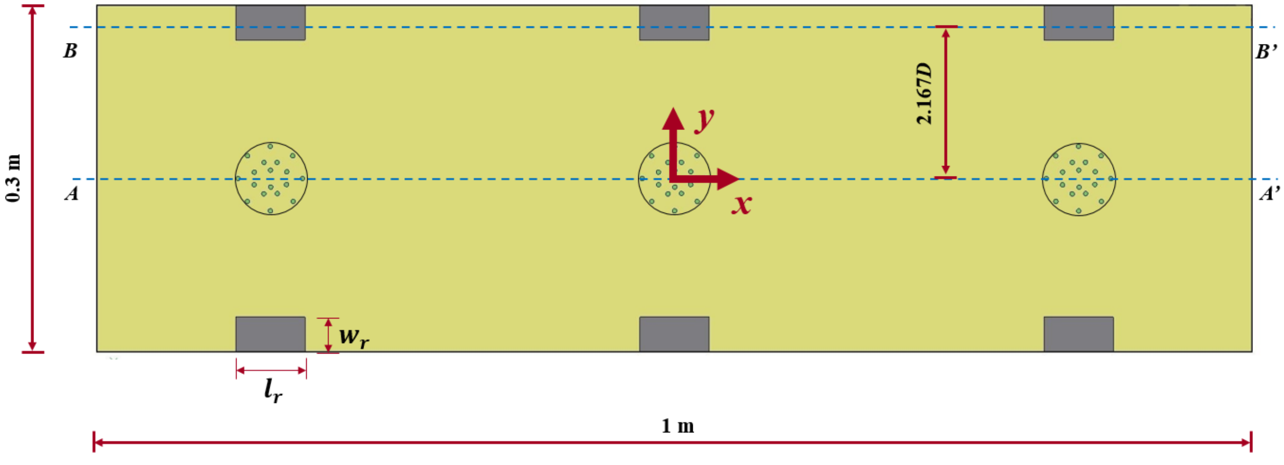


Figure 2: Schematic diagram of numerical model showing the geometrical details of the flow domain.

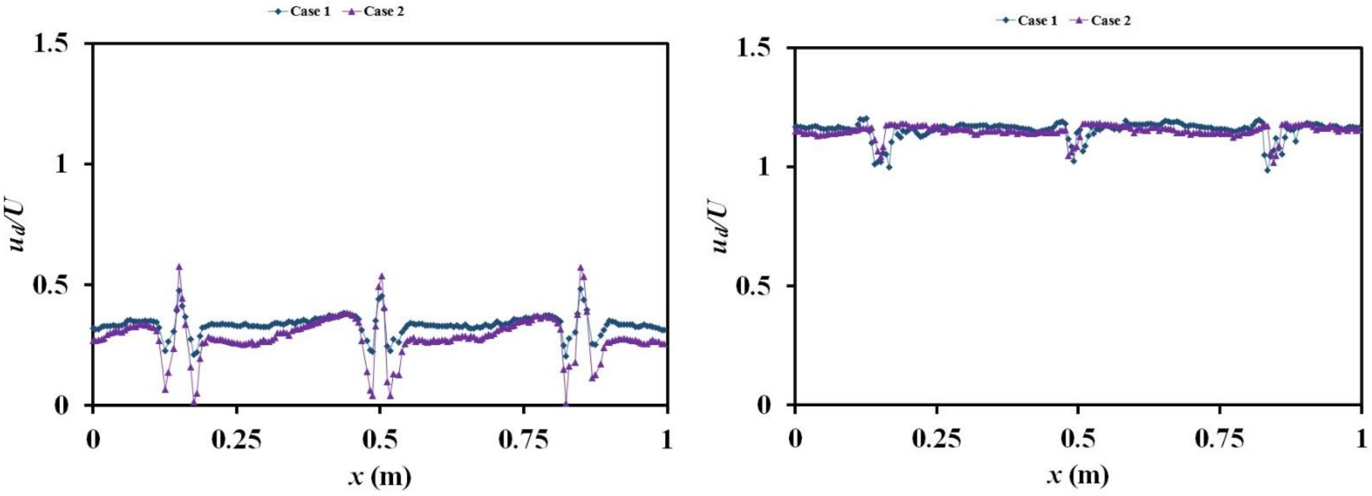
Table 2. Hydraulic Flow conditions of numerically simulated cases

Sr. No	***Q* (L/s)**	***H* (cm)**	**Vegetation Patch *D* (m)**	**Vegetation cylinder *d* (mm)**	**Roughness**		
					Height *k* (m)	**width w_r (m)**	**Length l_r (m)**
Case 1	18.01	13	0.06	4	0.03	0.03	0.06
Case 2	18.01	13	0.06	4	0.03	0.03	0.04

2 Results and Discussions

2.1 *Depth-Averaged velocities*

Numerically computed Depth-averaged velocities u_d were investigated at section *AA*' and *BB*' are shown in figure 3. The depth-averaged velocities were made dimensionless and normalized with the initial velocity *U* (*U* = *Q/A*). Section *AA*' passes through the origin i-e center of flow domain whereas section *BB*' is located at 2.167*D* in the transverse direction. From figure 3a distribution of depth-averaged velocities is highly non-uniform which illustrates that the flow in the vicinity of vegetated zone is highly non-uniform. The profile of u_d showed the rise in magnitude of velocities as the flow approached the vegetation patches and acquired a sawtooth distribution inside the vegetation patched regions. However, the decrease in magnitudes was observed in the flow zones just behind the vegetation patches. The reduction of velocities in the downstream of vegetated zones is due to the sheltering effects [14, 15]. At the upstream of vegetation patches, flow experiences a blockage and vegetation cylinders accelerated the flow. Due to the hindering effects of vegetation patch, an increase in depth-averaged velocities is rational. The alternate rise and fall in the distribution of u_d is due to the velocity difference between patched and wake zones. At section *BB*' magnitude of u_d is higher as compared to section *AA*'. However, a inflection instability in the velocity distribution was observed above the roughness elements. The reduction of the velocities above the roughness elements is due to acceleration of flow in lower zones. However, when the flow in low regions combines with the faster higher flow zones, a shear layer may develop which retards the flow slightly. The flow blockage in the case 1 is higher due to the small pitch to height ratio *P/K* at banks due to which the magnitude of u_d is slightly higher for case 1.



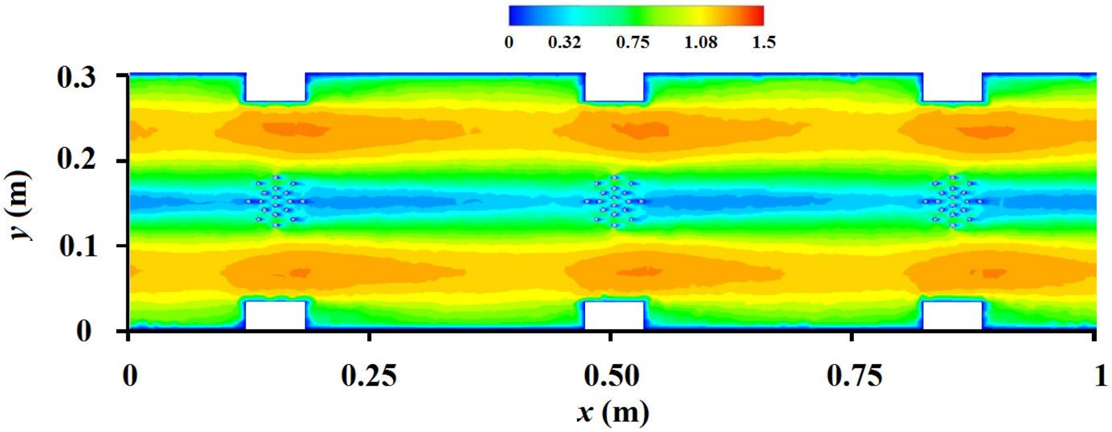
(a) (b)

Figure 3: Depth- averaged velocities at section (a). AA' (b) BB'

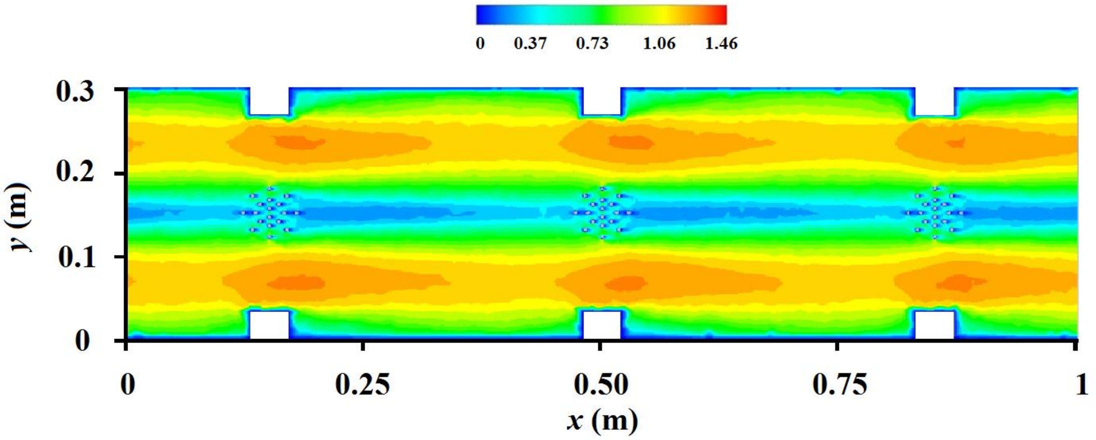
2.2 *Contours of Stream-wise velocity*

Figure 4 a-b shows the contours of stream-wise velocities along the horizontal plan at the z = 0.03 m. The longitudinal velocities are normalized with the average initial velocity U. The horizontal plan at z = 0.03 m passes through the top of Roughness elements. This surface was selected to study the flow resistance and wakes formed behind the vegetation patches and roughness elements. The flow zones located downstream of vegetation patches equipped with low velocity magnitudes. Meanwhile higher velocity regions can be observed adjacent to vegetation patches and roughness elements. When the flow approaches vegetation patches, it is diverted in the lateral direction away from the vegetated zones due to which a wake was formed downstream of circular patches having lower velocities. However, the increased hydraulic resistance accelerates the flow in regions adjacent to roughness elements and vegetation patches due to which noteworthy rise in magnitudes was observed. The interface of high velocity flow regions and low velocity zones are associated with the high shear stresses and scalar fluxes resulting in the lateral mass, oxygen defusal and concentration changes.

The flow zones downstream of vegetation patches supports the sediment depositions and ecological nourishments, which further enhanced the elongation and growth of vegetation mixture [14, 16]. A von Karman vortex street was also observed downstream of circular vegetation patches and is more elongated and prominent in case 1. The flow zones associated behind roughness elements are termed as the dead zones. The flow velocities were decelerated directly downstream of roughness elements. In the hydraulics engineering these dead zones have numerous advantages to promote the morphological diversity and enhanced ecological suitability for the nursery habitat and fish spawning [17, 18]. The overall magnitude of mean stream-wise velocity in case 1 is higher as compared to case 2. The increase in magnitude of velocities associated with both cases is 1.5 to 1.46 times that of the initial velocity.



(a)

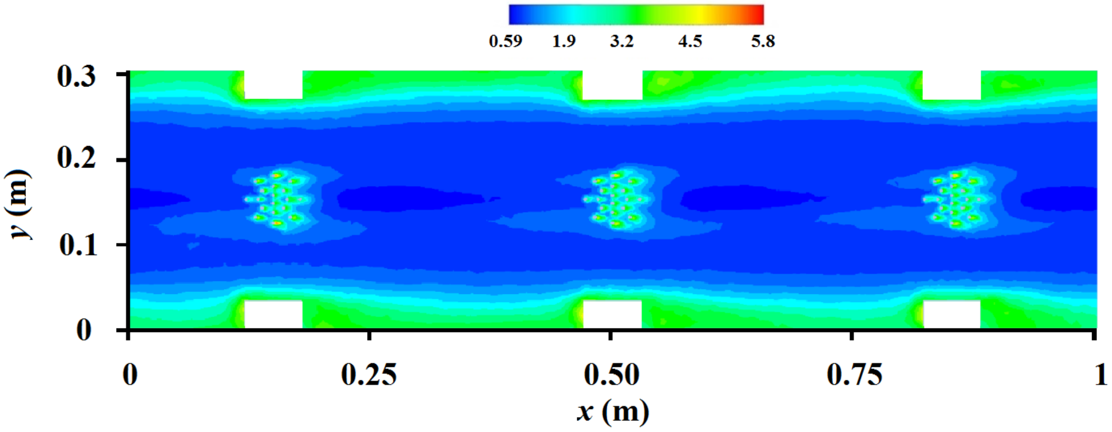


(b)

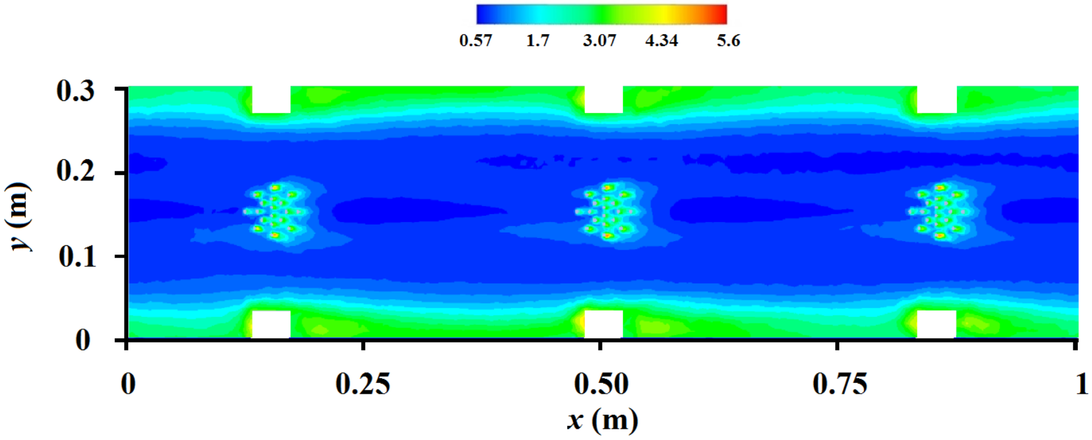
Figure 4: Contours of stream-wise velocity at z = 0.03 m (a). Case 1 (b) Case 2.

2.3 Turbulent Intensity

To quantify the turbulence in the flume turbulent intensity is investigated. Contour plots of turbulent intensity is investigated at the z = 0.03 m and shown in figure 5. A noteworthy difference was observed between the vegetated/roughness flow zones as compared to unobstructed free stream zones. Higher turbulences were observed in the vegetation patch zones and in the vicinity of roughness elements. The gradient of velocities in the dead zones and vegetated zones is higher due to which a larger turbulence was observed. Furthermore, the presence of wall in dead zones created a turbulence anisotropy which is also the responsible of higher percentage of turbulence at upstream and downstream of roughness elements. The distribution of turbulences in the vicinity of roughness elements flow zone is quite different as compared to flow zones associated at upstream and downstream of vegetation patches. The turbulence intensity is widely distributed and prolonged in the roughness elements flow zones, due to the turbulence anisotropy caused by the side walls. However, it is only confined in the vegetation patches and small magnitude is observed in the gaps located downstream of vegetation patches. The distribution of turbulent intensity in unobstructed free regions is uniform due to no effect of resistance. It can be seen in the figure that maximum 5.8, 5.6 % and minimum 0.59, 0.57 % of turbulent intensity has been observed for case 1 and 2 respectively. It can also be observed that size of roughness element has little contribution in the maximum turbulences in the dead zones.



(a)



(b)

Figure 5: Contours of Turbulent Intensity at z = 0.03 m (a). Case 1 (b) Case 2.

3 Conclusion

In the present study, a CFD technique has been used to investigate the flow features associated with the combined effect of bank roughness of two sizes and central emerged vegetation patches. This study shows the noteworthy effects of flow phenomenon in the channel comprising of two types of hydraulic roughness's.

- The distribution of depth-averaged velocities are small and uniform in the d/s of vegetation patches which shifted to highly non-uniform in the vegetation patches and acquired saw tooth distribution. Increased magnitude with inflectional instability in the distribution of depth-averaged velocity was observed directly above the roughness elements. The flow regions located d/s of any type of roughness have minimum flow velocities and supports deposition of sediments and wildlife habitat.
- Maximum 5.8 % and minimum of 0.57 % turbulent intensity was observed and mainly associated with the hydraulic resistance to flow. The turbulent intensity is maximum in the zones equipped with roughness elements and vegetation as compared to the free unobstructed regions.

4 Acknowledgement

The careful review and constructive suggestions by the anonymous reviewers are gratefully acknowledged.

5 References

[1] T. Meile, J-L. Boillat and A. J. Schleiss, "Flow Resistance Caused by Large-Scale Bank Roughness in a Channel," Journal of Hydraulic Engineering, Vol 137, pp. 1588-1597, 2011.
[2] A. Abbaspour and S. H. Kia, "Numerical Investigation of Turbulent Open Channel Flow with Semi cylindrical Rough Beds," KSCE Journal of Civil Engineering, Vol 18, pp. 2252–2260, 2014.
[3] H. Romdhane, A. Soualmia, L. Cassan and L. Masbernat, "Velocity profiles over homogeneous bed roughness," Fluid Mechanics Research International Journal, Vol 2, PP. 148-154, 2018.
[4] P. Orlandi, S. Leonardi And R. A. Antonia, "Turbulent channel flow with either transverse or longitudinal roughness elements on one wall," Journal of Fluid Mechanics, Vol 561, PP. 279-305, 2006.
[5] S. G. Sajjadi, and J. N. Aldridge, "Prediction of turbulent flow over rough asymmetrical bed forms," Applied Mathematical Modelling, Vol 19, pp. 139-152, 1995.
[6] J. Cui, V. C. Patel, C-L. Lin, "Large-eddy simulation of turbulent flow in a channel with rib roughness," International Journal of Heat and Fluid Flow, Vol 24, pp 372-388, 2003.
[7] J. Li and S. S. Li, "Near-bed velocity and shear stress of open-channel flow over surface roughness," Environmental Fluid Mechanics, Vol 20, pp. 293-320, 2020.
[8] H. Nepf, "Drag, turbulence, and diffusion in flow through emergent vegetation," Water Resources Research, 35 pp. 479-489, 1999.
[9] H. Nepf and M. Ghisalberti "Flow and transport in channels with submerged vegetation," Acta Geophysical, Vol 56, pp. 753-777, 2008.
[10] S. Wunder, M. Trevisson, C. Heckele, L. Chagot, B. Murphy, S. McLelland, F. Moulin and O. Eiff, "Near wake of emergent vegetation patches in shallow flow," River Flow 2018 - Ninth International Conference on Fluvial Hydraulics, Vol. 40, 2018
[11] C. Liu, Z. Hu, J. Lei and H. Nepf, "Vortex Structure and Sediment Deposition in the Wake behind a Finite Patch of Model Submerged Vegetation" Journal of Hydraulic Engineering, Vol. 44, DOI: 10.1061/(ASCE)HY.1943-7900.0001408.
[12] N. Anjum, U. Ghani, G. A. Pasha, M. U. Rashid, A. Latif and M. Z. Y. Rana, "Reynolds stress modeling of flow characteristics in a vegetated rectangular open channel," Arabian Journal for Science and Engineering, pp. 43, pp. 5551–5558, 2018.
[13] W. Li, D. Wang, J-L. Jiao and K-J. Yang, "Effects of vegetation patch density on flow velocity characteristics in an open channel," Journal of Hydrodynamics, Vol 31, pp. 1052–1059, 2019.

[14] F. Zhao and W. Huai, "Hydrodynamics of discontinuous rigid submerged vegetation patches in open-channel flow," Journal of Hydro-environment Research, Vol 12, pp.148-160, 2016.
[15] N. Anjum, U. Ghani, G. A. Pasha, G.A., A. Latif, T. Sultan and S. Ali, "To investigate the flow structure of discontinuous vegetation patches of two vertically different layers in an open channel," water, Vol 10, 2019.
[16] A. M. Gurnell, K. Thompson, J. Goodson and H. Moggridge, "Propagule deposition along river margins: Liking hydrology and ecology," Journal of Ecology, Vol 96, pp. 553–565, 2008.
[17] K. Uno, N. Kazuuma, G. Tsujimoto and T. Kakinoki, "Trapping effect of fine sediment in wand" Journal of Japan Society of Civil Engineers, Ser. B3 (Ocean Engineering), Vol 69, pp. 922–927, 2013.
[18] I. Nezu and K. Onitsuka, "PIV Measurements of side-cavity open-channel flows-Wando model in rivers," Journal of visualization, Vol 5, pp. 77–84, 2002.