NUMERICAL STUDY ON MIXED LAYER VEGETATION IN OPEN CHANNEL FLOW

HAMIDREZA RAHIMI
Engineering Department, Xi'an Jiaotong-Liverpool University, Suzhou, 215123, China.

XIAONAN TANG
Engineering Department, Xi'an Jiaotong-Liverpool University, Suzhou, 215123, China.

XANG WANG
Engineering Department, Xi'an Jiaotong-Liverpool University, Suzhou, 215123, China.

Abstract: Vegetation in river bed produces high resistance to flow and has a great impact on flow characteristics in rivers, especially during floods. The resistance due to vegetation in open channels reduces the flow discharge, which can lead to remarkable changes in physical and biological processes in aquatic environments. Numerical modelling analysis is undertaken for two and three layers vegetation in open channel flow. First, a model was developed to represent the contribution of vegetation in the RANs models through an additional momentum or resistance by modelling it as porosity domain. Then, the numerical models were tested and verified using the experimental data of Liu et al. (2008), and was then used to study for various other scenarios proposed. The modelling results by Ansys Fluent showed that the velocity profile is mostly uniform over the depth in both cases, except at location 1 for three layers vegetation. The increase of velocity profile above the short vegetation in three layers vegetation condition is much larger than that in two layers condition. Generally, in both cases the flow velocity inside the vegetation layer is significantly smaller than that in the surface layer (i.e. non-vegetation layer). A near-constant velocity dominates inside the vegetation layer, and then starts to increase near the interface at the top of vegetation. There is a sudden change in the shape of the velocity profile near the top edge of vegetation. The results also showed that for both two and three layers cases, the flow velocity is strongly dependent on locations, and that the distributions of the turbulent intensity attains maximum just around the edge of vegetation height.

Keywords: Velocity Profile, Emergent Vegetation, Submerged Vegetation, Turbulence Intensity, Ansys Fluent.

1 INTRODUCTION

The influence of riparian vegetation on ecological and flow process in channels has become an increasingly important aspect in river flood risk and environmental management. Many studies have been undertaken on the characteristics of the flow passed through the vegetation. These studies mainly focused on vegetation with the same height, which is not realistic in natural rivers and channels. There are only a few studies on flows with a mixing array of short and tall vegetation. In addition, most of the previous studies are just in either submerged or emergent flow condition; based on author's knowledge so far, little study has been undertaken in both conditions together. Most of the rivers and natural channels often have different height vegetation and are under both emergent and submerged condition together. Therefore, understanding the hydrodynamic characteristics of flow in vegetated open channels would provide valuable scientific basis for evaluating the effect of vegetation on river flows.

There is inherent difficulty in modelling flows under emergent and submerged conditions. Although the flow velocity distribution in vegetated open-channel has been studied experimentally and analytically in the literature, it cannot be done with a large variety of flows and vegetation configurations. In previous studies, logarithmic law or power-law distribution is well established to describe the velocity profile of the zone above the vegetation. However, in shallow flow conditions, the major portion of the flow above vegetation may be characterized more appropriately as a roughness sub-layer rather than an inertial sub-layer where logarithmic law 3 applies. Within a roughness layer, the local imbalance between production and dissipation of turbulence transport invalidates the logarithmic profiles (Nepf et al. 2007). There is a high demand to understand the effect of interfacial transport and turbulence transfer in scaling velocity profiles in vegetated open channel flows.

Although the flow velocity profile in vegetated open-channel has been studied experimentally and analytically in the literature, they are only with a limited range of flows and vegetation configurations.

Vegetation will change the velocity distributions and some other characteristics of flow. Previous experimental studies may be classified into two groups. One group of researchers conducted real and natural vegetation as experimental material, and the other used rigid cylinders to simulate vegetation (Tsujimoto and Kitamura 1990; Nepf 1999; Carollo, Ferro, and Termini 2002; Nezu and Sanjou 2008).

Based on Nezu and Nakagawa's studies, in normal open channel flows without any vegetation, vertical velocity profile is logarithmic (Te Chow 1959; Nezu and Nakagawa 1993).

Nevertheless, the vegetation generates resistance, induces drag forces, and can cause significant changes in velocity profile. Vegetation in the channel can be either emergent or submerged. In emergent condition, the velocity profile can be uniform over the flow depth (Tsujimoto and Kitamura 1990; Stone and Shen 2002). In submerged condition, the velocity profile followed an S-shaped pattern (Kouwen, Unny, and Hill 1969; Temple 1986; Ikeda and Kanazawa 1996; Carollo et al. 2002).

Stone and Shen (2002) found that the flow under submerged condition is often more complicated than that in the emergent condition, and they also pointed out that the velocity in the surface layer is substantially larger than that in the vegetation layer. Recently Liu et al. (2008) undertook some experiments with rigid cylinders and measured the velocity at different locations within the vegetation area using LDV (Laser Doppler Velocimetry). They noticed that in both submerged and emergent conditions, the velocity through vegetation is dependent on vegetation spacing and density. In denser areas, the velocity is different at different locations and depth, but in low-density vegetation, the velocity varies remarkably in different locations. Based on their experiments, velocity inside the vegetation layer changes small vertically, followed by rapid increase near the interface close to the top of vegetation. In this research, experimental by Liu et.al (2008) have been modeled by Ansys Fluent and after the calibration other experimental cases have been studied.

2 METHODOLOGY

Considering that the aim of this study is to solve the flow field around cylinders in a staggered arrangement, the flow around the cylinders is modeled by ANSYS-Fluent, and then the results are compared with experimental data to verify the validity and accuracy of the discretization and boundary conditions and other important parameters used in the modelling. Liu et al. (2008) did some experiments under different conditions and configuration in a 4.3 m long and 0.3 m wide flume at the Baker Environmental Hydraulic Laboratory of Virginia Tech. The vegetation was simulated by acrylic dowels with 6.35 diameter and 76 mm height. In our study of submerged single layer, one of the Liu experimental tests with 6 different locations was considered as validation purpose (Figure 1). They measured the velocity profile for z/h_s 1.5-1.57 at multiple locations by using LDV, where z and h_s represent the depth and height of short vegetation respectively, and found that the velocity is nearly constant and increases significantly at the top of the vegetation. They also reported that in the region immediately behind dowels, the highest turbulence intensities exists and is unstable, especially at the dowel edges, where significant mass and momentum exchange occurs.

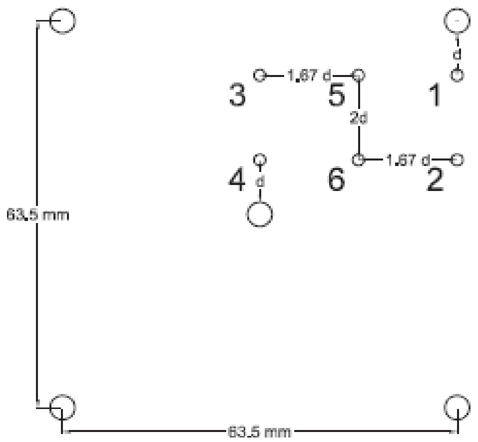


Fig.1Dowel arrangement and measurement location in Liu et al. experiment.

In this paper, a 3D model was created using ANSYS Space Claim and the four boundary types were velocity inlet, pressure outlet (outlet and upper section of the model), symmetry and wall. Inlet velocity was set to 0.327 m/s and outlet pressure was set to 0, which represents the atmospheric pressure (operating pressure) of zero (Figure 2).

Given the inlet velocity of 0.327 m/s and hydraulic diameter of 0.88 m, the flow Reynolds number is calculated to be 37300, so the flow is considered fully turbulent. Thus, flow turbulence was modeled using k-ε empirical equations and von-Karman standard wall functions were used to capture gradients of velocity in the vicinity of viscous sub layer.

In order to obtain the mesh-size independent results, three meshes with different sizes were generated in ANSYS-ICEM CFD. The mesh sensitivity results from cell sizes were analyzed, and the mesh chosen meets the condition that changing the sizes of the network does not very much change the results. Therefore, a generated mesh with 1,385,047 cells was used to study the flow characteristics throughout the numerical analysis (Figure 3). The finer grid has little effect on the velocity distribution and turbulent intensity of the flow. Thus, the results are independent of the grid sizes. Since the velocity gradient is very large in the direction perpendicular to the wall around the cylinder, a boundary layer mesh was used close to the wall and the size of the first cell at the boundary of the cylinder wall was assigned to be 0.0014 m.

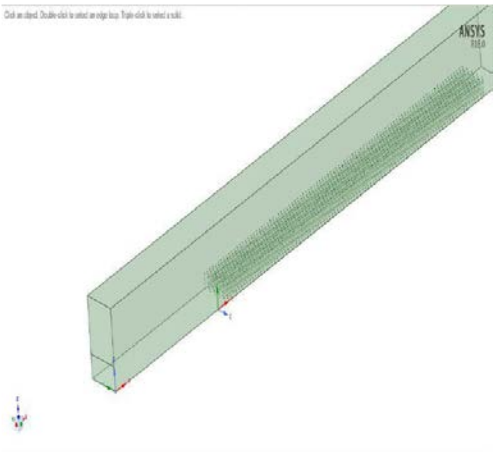


Fig.2. Model designed by Space Claim

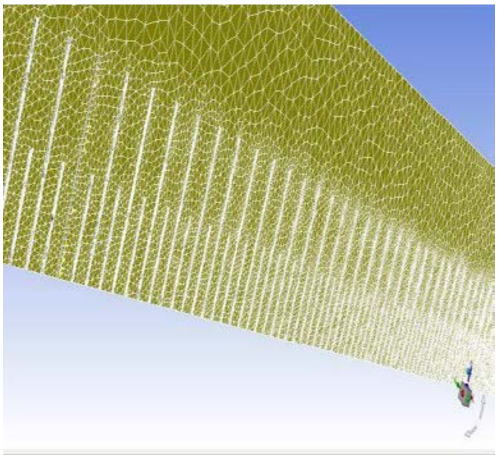


Fig.3. Model created by Space Claim

For conditions at the inlet or outlet boundaries where the fluid enters or leaves the computational domain, their boundaries are usually defined as the condition at a specified velocity or with a specific pressure. Due to the pairing of the pressure and velocity in the momentum equations, only one of the two conditions of pressure or velocity is defined. For the inlet boundary condition, the velocity value is determined along x coordinate with U = 0.327 m/s, and for the outlet boundary condition, the static pressure is determined through the outlet, which is the same atmospheric pressure as the relative pressure is zero.

For the turbulence parameters in the inlet and outlet, the following correlations are used based on Ansys Fluent user's guide:

$I=0.16(Re)^{1/8}$ (1)

$\ell=0.07D$ *(2)*

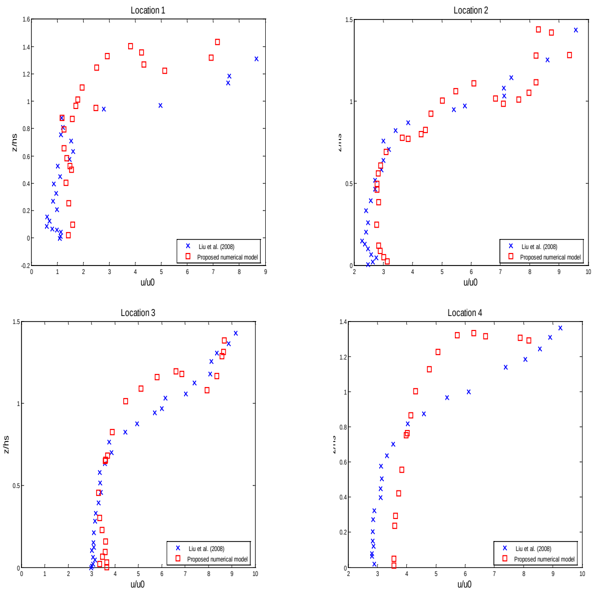
where I represents the turbulent intensity defined as I= $\acute{u}_i/u_i$, and ℓ is the turbulence length scale which indicates the size of the large vortices in the turbulent flow, and D is the diameter of dowel. The value of I was taken 5% in the modelling.

For dowels arrangement, a wall boundary condition was used. Since the fluid does not cross the wall, the component of the velocity normal to the wall is zero. In addition, due to the no-slip wall condition, the tangential component of the velocity is also zero. In the top side boundaries, the pressure is considered atmospheric and relatively 0. In order to simplify the current modelling, the Symmetry boundary condition was used (no-shear wall) at the right side of the model.

In the present analysis, water (an incompressible fluid) with a density $\rho = 1000\ kg/m^3$ and the dynamic viscosity $\mu = 0.00104$ was used. The standard k-ε turbulence model was used. The SIMPLE algorithm was chosen to solve the governing equations with second order upwind discretization. For convergence of solutions, the model is first solved in the steady state, then solved in the unsteady state and time-variant with a time step size of 0.05s and the number of time steps of 700 to 1000.

3 RESULTS AND DISCUSSION

The modelling results are analyzed through the CFD-Post component. The results are shown in Figures 4 and 5 for the instantaneous velocity and turbulence, respectively, in proposed locations.



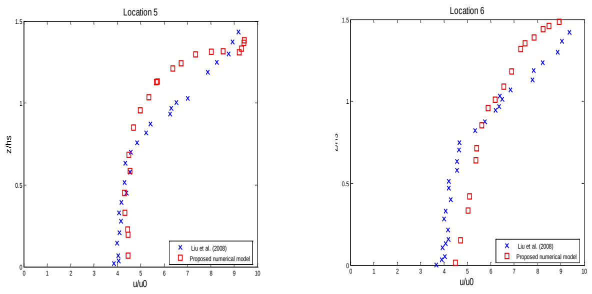


Fig.4. Velocity profile comparison of location 1 to 6 for
Liu et al. (2008)'s experiment. uo= shear velocity.

Figure 4 indicates that the overall trends of velocity profile through vegetation show an acceptable agreement, although there are some differences in the vegetation region in location 1, 4 and 5, which may be due to errors from the experiment or modelling. The location immediately downstream of a dowel (location 1) experienced a spike in velocity near the bed of channel and stayed almost constant through a certain depth of flow, then followed by a rapid increase in velocity at top of the cylinders. Although there are some fluctuation in the trend of modelling results but almost I all of locations modelling trend's shape are almost as the experimental data. The location 5 and 6 (both located in the open region) had the highest velocity compared with those in other measurement locations such as 1 and 4 in behind and front of dowels respectively. These results indicate that the presence of cylinders has remarkable effect on velocity.

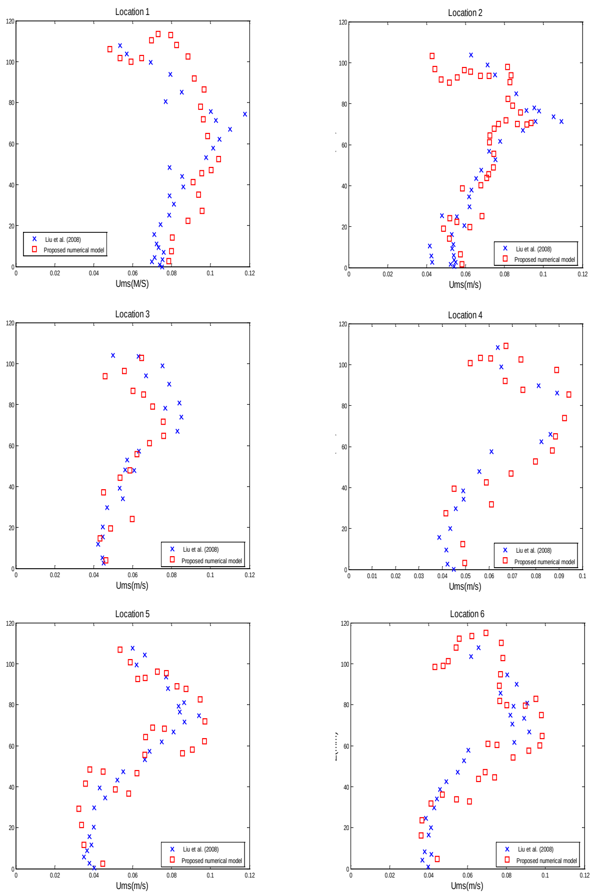
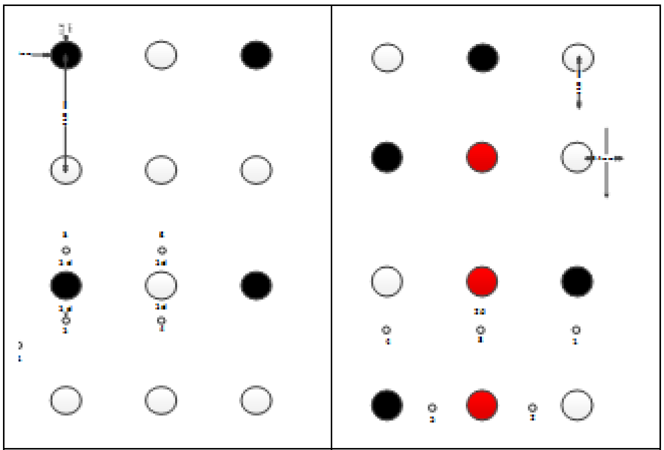


Fig.5. Turbulence intensity profiles of location 1 to 6 for
Liu et al. study and proposed model.

Figure 5 shows the comparison of turbulence intensity profiles at location 1 to 6. Generally, the modelling results agree reasonably well with the experimental data. The turbulence intensity is almost constant near the bed of the channel and rises slightly to reach a peak where the flow is highly sheared near the edge of vegetation at $z/h_{s\,=\,0.8\text{-}1}$. Above the dowel height the turbulence intensities have been declined and in some cases like location 1 even reached less than the initial level. Generally although there are some marked differences in some points like some points of location 1, but in most of the locations the proposed model matched by experimental data which approved the process of modelling.

3.1 Comparison of mixed layers vegetation by numerical modeling

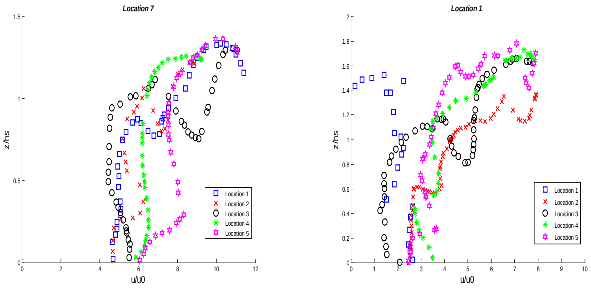
After the validation, the numerical model by Ansys Fluent was used to evaluate the flow parameters such as velocity and turbulence intensity when the flow passed through vegetation in two and three layers vegetation. The vegetation arranged in a unique pattern to simulate the vegetation based on staggered formation.
Figure 6 shows the formation of cylindrical dowels and measurement locations. Figure 6a shows two layer vegetation by two different types of dowels. The vegetation was simulated by 6.35 mm diameter cylindrical dowels of two heights in 100 mm and 200mm. In Figure 6b a medium-sized dowel of 150 mm height was added and thus three different types of dowels were used to simulate three layers vegetation and study the effect of dowel's type and density on flow structure.



a b

Fig.6. Dowel arrangement for two and three layers vegetation. The black, red and white circles represent tall, medium and short dowels respectively. The arrow shows the flow direction. Small circles (o) represent the measurement points by ADV.

Five measurement locations were carefully chosen in order to provide comprehensive data about the flow characteristics around different types of dowels and in the regions without any dowels. Figure 7.a illustrates the velocities at different locations in a case with two layers vegetation, and indicates that almost a constant velocity dominates in a certain height in the vegetation layer and increases near the interface. Almost same trend has been happened for three layers vegetation except location 1 in which there has been a declined all over the depth after the short dowel height. However, in both mixed layer conditions (two and three dowels), the velocity depends on the location of measurement as well as vegetation density. In two layers vegetation the velocity difference is much more observable in different locations (Fig. 7a). In three layers vegetation, the velocity rises to near the top of short dowels and in fully submerged condition in all other locations except location 1 (Fig. 7b). Generally, in both cases there is abrupt change in the shape of the velocity profile near the top of the short dowel.

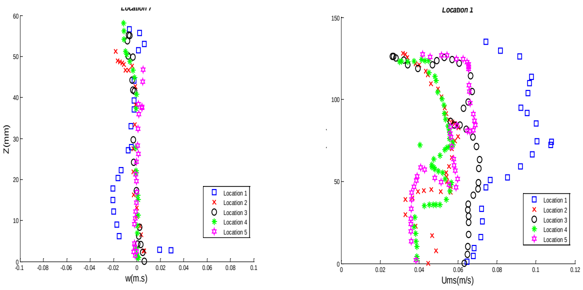


a - two layers b – three layers

Fig.7. Velocity profiles by model for three types of dowels. u_o = shear velocity and h_s =short dowel height.

Furthermore, in two layers vegetation (Figure 7a), the velocity profile below the top of short dowels is small but gradually increases as increasing flow depth. In the layer above the short dowel height, the trend experienced swift increase.

The velocity profiles in different locations in three layers vegetation (Figure 7b) shows that locations 2 and 5 in the open stream region have the largest velocity while the velocity are relatively small in the locations that are immediately behind the dowels. Location 1 behind the tall dowel has a velocity profile, which is a completely different pattern from those in other locations, and this indicates that the flow behind the dowel may have much impact by the dowel due to the eddy/wake effect, and this phenomenon further investigation.



a – two layers b – three layers

Fig.8. Streamwise turbulence intensity for the three layers vegetation.

Figure 8a and 8b indicate streamwise turbulence intensities in two and three layers vegetation respectively. Based on the modelling data the turbulence intensity attains its maximum value at around the top of vegetation. In the two layers vegetation case, the dependence of turbulence to height of the vegetation and measurement locations is much less than that in the three layers vegetation case.

4 CONCLUSIONS

Numerical modelling was undertaken for the vegetated flow, where the vegetation was modeled in two different conditions of two and three layers vegetation. Various locations were selected to cover the flow conditions for both cases. To be more specific, the measurement locations was carefully chosen in all important sections of the vegetation region, such as open stream location, ones behind and in front of different height dowels.

The paper compared velocity profiles and turbulence intensities for depth-limited open channel flows with different mixing layer vegetation. The modelling results by Ansys Fluent showed that the velocity profile is mostly uniform over a certain depth near the channel bed in the both cases, except location 1 for three layers vegetation. The increase of velocity profile above the short vegetation is much larger in three layers vegetation condition than that in two layers condition. Generally, in both cases the flow velocity within the vegetation layer is significantly smaller than that in the surface layer. A near-constant velocity dominates inside the vegetation layer, and then starts to increase rapidly near the top of short vegetation. There is a sudden change in the shape of the velocity profile near the top of short vegetation. The results also showed that for both two and three layers cases, the flow velocity is strongly dependent on locations. In addition, the distributions of the turbulence intensity attain maximum just around the top of vegetation.

REFERENCES

[1] ANSYS, Inc. (2018), "ANSYS Fluent User's Guide, Release 19.0", Equation (6.68).

[2] Carollo, F., et al. (2002). "Flow velocity measurements in vegetated channels." Journal of Hydraulic Engineering 128(7): 664-673.

[3] Ikeda, S. and M. Kanazawa (1996). "Three-dimensional organized vortices above flexible water plants." Journal of Hydraulic Engineering 122(11): 634-640.

[4] Kouwen, N., et al. (1969). "Flow retardance in vegetated channels." Journal of the Irrigation and Drainage Division 95(2): 329-344.

[5] Liu, D., et al. (2008). "An experimental study of flow through rigid vegetation." Journal of Geophysical Research: Earth Surface 113(F4).

[6] Nepf, H. (1999). "Drag, turbulence, and diffusion in flow through emergent vegetation." Water resources research 35(2): 479-489.

[7] Nepf, H., et al. (2007). "Transport in aquatic canopies." Flow and Transport Processes with Complex Obstructions: 221-250.

[8] Nezu, I. and H. Nakagawa (1993). "Turbulence in open channels." IAHR/AIRH Monograph. Balkema, Rotterdam, The Netherlands.

[9] Nezu, I. and M. Sanjou (2008). "Turburence structure and coherent motion in vegetated canopy open-channel flows." Journal of hydro-environment research 2(2): 62-90.

[10] Stone, B. M. and H. T. Shen (2002). "Hydraulic resistance of flow in channels with cylindrical roughness." Journal of Hydraulic Engineering 128(5): 500-506.

[11] Te Chow, V. (1959). Open channel hydraulics, McGraw-Hill Book Company, Inc; New York.

[12] Temple, D. M. (1986). "Velocity distribution coefficients for grass-lined channels." Journal of Hydraulic Engineering 112(3): 193-205.

[13] Tsujimoto, T. and T. Kitamura (1990). "Velocity profile of flow in vegetated-bed channels." KHL progressive Report 1.