ARPN Journal of Engineering and Applied Sciences
©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

USE OF CFD TO MODEL EMERGENT VEGETATION IN DETENTION PONDS

Alexandros Tsavdaris, Steve Mitchell and John B. Williams
School of Civil Engineering and Surveying, Portland Building, Portland Street, University of Portsmouth, Portsmouth, United Kingdom
E-Mail: alexandros.tsavdaris@port.ac.uk

ABSTRACT

This paper investigates the accuracy, applicability, and suitability of two different numerical modelling approaches available in Ansys Fluent 12.1 for the study of flow in detention ponds with emergent vegetation by making use of experimental results obtained in a laboratory flume. The aim of this investigation is to formulate an automated first-order approximation technique that could be used as part of an urban drainage model; such an approach could be an accurate yet practical technique for modelling the effects of vegetation in ponds at pre-construction stage in the interests of predicting general flow patterns. Using the actual vegetation density of a surface water detention pond located at Waterlooville, Hampshire, UK, replicated in a laboratory flume, two different Computational Fluid Dynamics modelling strategies were tested. The first involved the specification of the individual stems within the computational domain, and these results showed very good agreement with experimental data. In the second approach, a porous zone condition was applied in the vegetated region, and here the results seem to be appropriate for predicting general flow arrangements, though without being hydro-dynamically as accurate as the first approach.

Keywords: fluid mechanics, detention ponds, vegetation, research and development, physical modelling.

1. INTRODUCTION

The presence of vegetation in detention ponds may result in patterns of flow that differ from those in non-vegetated systems. Predictions of the flow regimes involved are useful for designing ponds for the safe disposal of polluted highway runoff as part of a sustainable drainage system, for example. Vegetation has a significant impact on the hydraulic behaviour and flow structure of open channel systems. The existence of vegetation within the flow domain tends to increase the hydraulic resistance via turbulence and drag (Choi and Kang, 2006). Turbulence increases the mixing within the watercourse, thereby influencing the conveyance routes of contaminants and suspended solids in the water. At the same time, vegetation reduces the discharge capacity and causes flow resistance, which needs to be studied and understood in the interest of flood safety (Fu-sheng, 2008). According to Ghao *et al.* (2011), there are three different types of vegetated open channel flows, according to the height of vegetation (h_p) relative to the total water depth (H), namely terrestrial canopy flows ($h_p/H \approx 0$), flows with submerged vegetation ($0<h_p/H<1$), and flows with emergent vegetation. There have been many studies of the effect of vegetation on flow, both experimental (Järvelä, 2002; Feng-feng, 2007; Fu-sheng, 2008; Er-qing and Xing-e, 2010) and/or utilizing numerical modelling (Jian-tao, 2008; Li and Zeng, 2009; Pei-fang and Chao, 2011; Larmaei and Mahdi, 2012; Mattis *et al.*, 2012). However, there is still much work to be done in understanding the hydraulics of vegetated flows in pond systems and interpreting that knowledge into controllable management methods (Folkard, 2011).

Progress in efforts to develop a practical and widely applicable method of predicting flow resistance in vegetated channels has been difficult due to the lengthy list of variables that must be taken into account (drag coefficients, roughness factors, projected area, Reynolds number, channel slope, plant height, etc.). Furthermore, the breadth of the viewpoints of scientists and engineers studying vegetated flows has led to a plethora of different research philosophies (Folkard, 2011). The Chezy, Darcy-Weisbach and Manning's equations, each of which uses a roughness coefficient to quantify flow resistance, are the most widely-used formulas in terms of depicting vegetated flows (Hamill, 2001). Naden *et al*. (2006) however argued that the use of Manning's n is not appropriate for many vegetated flows, especially where emergent vegetation is present. In addition, despite the proposals of generalized forms of equations for predicting flow structure (Jordanova *et al.*, 2006; Pei-fang and Chao, 2011; Wen-xin *et al.*, 2012) and porosity-based flow modelling (Stovin *et al*., 2009; Saggiori, 2010), Manning's n is still the most common way of characterizing flow resistance (Folkard, 2001). This observation is unexpected given the fact that Manning's n lacks theoretical cogency and often produces inaccurate results in practice (Green, 2005; Foklard, 2011). Green (2006) contends that a cross-sectional blockage factor, used in conjunction with measurements of the decrease in hydraulic radius (caused by vegetation), provides the most accurate prediction of flow resistance. Nonetheless, the measurement of multiple cross-sections is impractical and logistically prohibitive with respect to the operating standards of hydraulic engineers. Moreover, the current methods adopted by hydraulic engineers for flow structure predictions in vegetated domains are idiosyncratic and imprecise (Folkard, 2011). Additionally, the effectiveness of theoretical and empirical approaches is limited by the complexity of natural systems, making the adaptation of results (from idealized configurations) to practical

ARPN Journal of Engineering and Applied Sciences
©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

problems extremely difficult. In addition, this route is limited by logistical constraints due to the need for time-consuming detailed and extensive field measurements.

A well-established means of evaluating the impact of vegetation on flow is via the use of Computational Fluid Dynamics (CFD). The use of CFD can be cost-effective, and compared to traditional approaches, a less time-consuming technique for assessing flow in vegetated domains (Liwei *et al.*, 2008). However, due to the difficulty of designing individual plants within the computational domain and the time consuming development of user defined numerical models that calculate the drag force exerted by the vegetation as discussed above, another approach could be either the numerical treatment of the vegetation as a porous zone, or the development of micro-scale models as suggested by Mattis *et al.* (2012). Using experimental data from Schucksmith (2008), Saggiori (2010) evaluated the effect of vegetation on flow within a pond constructed to control storm water runoff. She used the porous zone feature of Ansys Fluent (Ansys®, 2009) to replicate vegetation and reported no significant impact regarding velocity reduction in terms of vegetation. However, Yan (2011) noted that the literature describing experiments on turbulent flows in porous media was very limited, thereby casting doubt on the modelling accuracy of such cases. As a result, information on the applicability of porous zones (Ansys®, 2009) to investigate the effect of vegetation on flow in detention ponds is scarce.

In light of the foregoing, this study focuses on the development of an automated first-order approximation procedure regarding the effect of emergent vegetation on flow in ponds by use of CFD. This method is directed to pond designers who wish to examine accurately and rapidly the flow patterns of vegetated detention ponds without requiring the development of unfeasible numerical models and logistically challenging field measurements. This specific modelling technique examines the effect of vegetation on flow by employing two different strategies. The first involves the replication of the vegetation elements within the computational domain and the second strategy makes use of porous zones. Ansys Fluent 12.1 (Ansys®, 2009) incorporates a porous media condition that introduces flow resistance, based on empirically derived coefficients, but there have been few instances of its use to model vegetation in ponds (Stovin *et al.*, 2009; Saggiori, 2010). Consequently, this paper aims to investigate the performance and accuracy of these two different CFD modelling strategies on the basis of a flume study, in the interest of developing a practical approach for modelling vegetation in ponds. It should be noted that this modus operandi is only applicable to a specific type of vegetation (typically reeds), which is predominantly used in the construction of detention ponds. The model was validated by comparing measured and modelled depth-averaged velocity. This is a relatively accurate, time-effective and cost-effective approach.

2. MATERIALS AND METHODS

2.1. Experimental setup

The vegetation cover (VC) of a detention pond used to manage road runoff, located at Waterlooville, UK (Latitude = 50.881315, Longitude = -1.037575) was surveyed on the 31st of January 2012 (in the winter reeds have no leaves, making the replication closer to reality; further, most storm events occur during winter and autumn) using quadrats with an area A=0.5 m². The survey included 20 random sampling points in the shallow part (H<1m) of the pond and another 20 in the deep part (H>1m). Two different populations were identified. All statistical results were obtained using the Minitab™ software (Minitab, 2012). For the shallow region, the survey revealed median values of 186 *Phragmites australis* (P.A) and 20 *Typha latifolia* (T.L) per m² (VC_S). For the deep region, the survey indicated median values of 45 P.A and 22 T.L per m² (VC_D). In addition, the survey showed a median plant diameter (D_P) of D_p=0.01 m for P.A and D_p=0.035 m for T.L. On the basis of the survey data, two different vegetation patches were constructed in the flume at densities of VC_S (or) VC_D. The effect of each patch on flow was examined separately. Emergent vegetation was simulated using stiff bamboo sticks with identical diameters to those observed, with height $h_p \approx$ 0.35 m. The vegetation was glued (Durostick® waterproof PVC adhesive) to a PVC plate with thickness (z) of 0.015 m that covered the whole width of the flume (W=0.290 m), after the plate had first been pre-drilled with holes for the bamboo sticks. Each stick was placed and glued individually using a staggered arrangement to mimic real conditions. The patches were left to dry for 24 h in order to ensure the adhesive had properly set. The vegetation patch had a length (L) of 1.0 m and was placed with its leading edge 1.5 m downstream of the inlet. Two other PVC plates were also attached to the bed (upstream and downstream the vegetation patch), each with z=0.015 m and L=1.5 m, in order to ensure uniform flow conditions near the bed.

Experiments were conducted in the hydraulics laboratory of the University of Portsmouth using a calibrated recirculating flume with L=6 m and W=0.29 m (Figure-1), with maximum H=0.3 m. Steady non-uniform flow conditions were determined prior to the experiment. The bed of the flume is made of steel and the side walls are made of glass. The flow rate was measured using an analogue discharge meter and a downstream weir was used to establish uniform flow. The bed slope was set to 0.4% as for the detention pond, and the height of the downstream weir was not adjusted at any point. Flow depth was measured using a point gauge mounted to a movable carriage attached to rails on top of the flume. A calibrated Vale port Model 801 electromagnetic flow meter was also attached to measure the time-averaged fluid velocity (U_T) in the x direction (sampling time=30 s) at each point of interest. These measurements were made at 0.5 m intervals on the horizontal x axis and every 0.01

ARPN Journal of Engineering and Applied Sciences
©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

m (for odd H) or 0.02 m (for even H) on the vertical z axis.

The depth-averaged fluid velocity in the x direction (U_{exp}) was calculated for each position from the measured velocities. The experiment was performed under steady flow conditions for two different flow discharges, Q_1= 0.0077 (m³/s) and Q_2= 0.0174 (m³/s). All the velocity measurements took place on the centreline of the flume. A total of three different experimental conditions were used namely no vegetation (NV EXP.), shallow-water vegetation (S EXP.) and deep-water vegetation (D EXP.). The variable of depth could not be evaluated due to the size of the available flume apparatus. For this experiment, only the effect of VC on different flow conditions could be assessed.

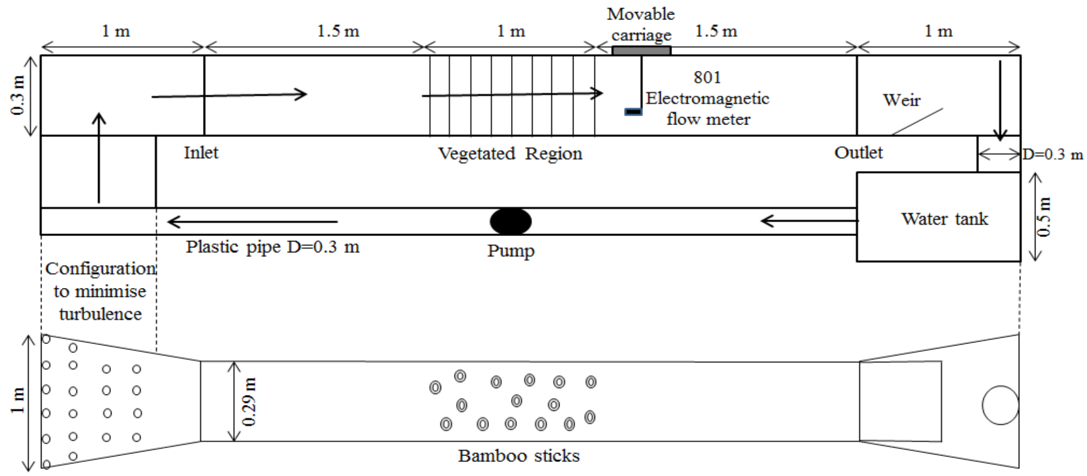

Figure-1. Longitudinal section and plan view of the flume.

2.2. CFD model development

The initial stage in the development of the CFD model was the construction of the computational grid (see Table-1) using the Geometry and Mesh Building Intelligent Toolkit (GAMBIT) software (Ansys®, 2009). All cases had a mesh quality of 1 (on a scale from 0-1; where 0 corresponds to low quality mesh and 1 to high quality mesh). The 3D Reynolds-Averaged-Navier-Stokes (RANS) equations for steady, incompressible flow in combination with the realizable k-ε turbulence model (Shih *et al*., 1995), for calculating the turbulent stresses, were solved using the Ansys Fluent 12.1 CFD code (Ansys®, 2009). Yan (2011) indicated that the realizable k-ε model is among the most accurate of all the available turbulence models. All the numerical schemes behind the calculation process can be found in the software user's guide (Ansys®, 2009) and are omitted here for brevity. The model solves the governing non-linear and coupled equations sequentially, thus several iterations of the solution loop must be performed before the minimum convergence criterion is fulfilled (reduction of 10^3 order magnitude of the scaled residuals from the continuity, momentum and turbulence equations; Ansys®, 2009). CFD codes employ a control volume technique to convert the governing equations to algebraic equations that can be solved numerically. The integration of the governing equations yields discrete equations that conserve each quantity for each control volume (Souliotis and Prinos, 2011). All equations were discretised using the second order upwind scheme (Ansys®, 2009). The SIMPLE algorithm was used for pressure-velocity coupling and the Least-Squares-Cell-Based method was used for the evaluation of gradients (Ansys®, 2009). For this study the most robust boundary conditions were applied as suggested by Ansys® (2009) and Souliotis and Prinos (2011). The boundary conditions for the inlet and outlet were velocity inlet and outflow respectively. The free surface was modelled as a symmetry boundary condition and the walls (bed, side) had a no slip condition (adiabatic wall) and a roughness height of 7 x 10^{-4} m as in Souliotis and Prinos (2011). A uniform velocity was assigned to the inlet.

The porous media model integrates an empirically determined flow resistance in a cell zone of the model defined as "porous". Porous media are modelled by the addition of a momentum source term to the standard fluid flow equations. According to Ansys® (2009), the source term is composed of two parts: a viscous loss term (Darcy, the first term of equation (1)) and the inertial loss term (the second term).

$$S_i = -\left(\sum_{j=1}^{3} \mathbf{D_{ij}} \mu v + \sum_{j=1}^{3} \mathbf{C_{ij}} \rho |v| v_j \right) \quad (1)$$

where S_i is the source term for the ith (x, y, z) momentum equation, μ is the viscosity, ρ is the fluid density, $|v|$ is the magnitude of the velocity, and D_{ij} and C_{ij} are prescribed matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity in the cell. For a simple homogenous porous media

©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

$$S_i = -\left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i\right) \tag{2}$$

where α is the permeability and C_2 is the inertial resistance factor. In laminar flows through porous media, the pressure drop is typically proportional to velocity and the constant C_2 can be considered to be zero. Ignoring convective acceleration and diffusion, the porous media model then reduces to Darcy's law

$$\nabla p = -\frac{\mu}{\alpha} \vec{v} \tag{3}$$

At high flow velocity, the constant C_2 provides a correction for inertial losses in the porous medium. This constant can be viewed as a loss coefficient per unit length along the flow direction. Dropping the permeability term yielding the following simplified form of the porous media equation

$$\nabla p = \sum_{j=1}^{3} C_{2ij} \left(\frac{1}{2} \rho v_j |v|\right) \tag{4}$$

At turbulent flows, packed beds (vegetated regions) are modelled using both permeability and an inertial loss coefficient. In order to derive the appropriate constants the Ergun equation must be used (Ansys®, 2009). The Ergun equation is a semi-empirical correlation applicable over a wide range of Reynolds number and for many types of packing.

$$\frac{\Delta p}{l} = \frac{150\mu(1-\varepsilon)^2}{D_p^2 \varepsilon^3} v_\infty + \frac{1.75\rho(1-\varepsilon)}{D_p \varepsilon^3} v_\infty^2 \tag{5}$$

In equation (5) Δp is the pressure loss, μ is the viscosity, D_p is the mean particle diameter, l is the bed depth and ε is the void fraction, defined as the volume of voids divided by the volume of the packed bed region. Comparing this equation with Darcy's law in porous media and Inertial Losses in porous media, the permeability Equation (6) and inertial loss coefficient Equation (7) in each component direction may be identified as

$$\alpha = \frac{D_p^2}{150} \frac{\varepsilon^3}{(1-\varepsilon)^2} \tag{6}$$

$$C_2 = \frac{3.5}{D_p} \frac{(1-\varepsilon)}{\varepsilon^3} \tag{7}$$

Furthermore, Ansys Fluent identifies a variable known as turbulent intensity (Ansys®, 2009):

$$I = 0.16 * Re^{(-0.125)} \tag{8}$$

where Re is the Reynolds number. This particular variable indicates how turbulent the flow is; e.g., Re=50000 results in approximately I = 4% (Ansys®, 2009). The two modelling strategies include:

- A CFD model replicating the IS with identical arrangement of the two (shallow and deep) experimental vegetation patches (Cases 3, 4, 7, 8) (Figure-2)
- A CFD model replicating the vegetation patches using the porous zone option (Cases 5, 6, 9, 10)

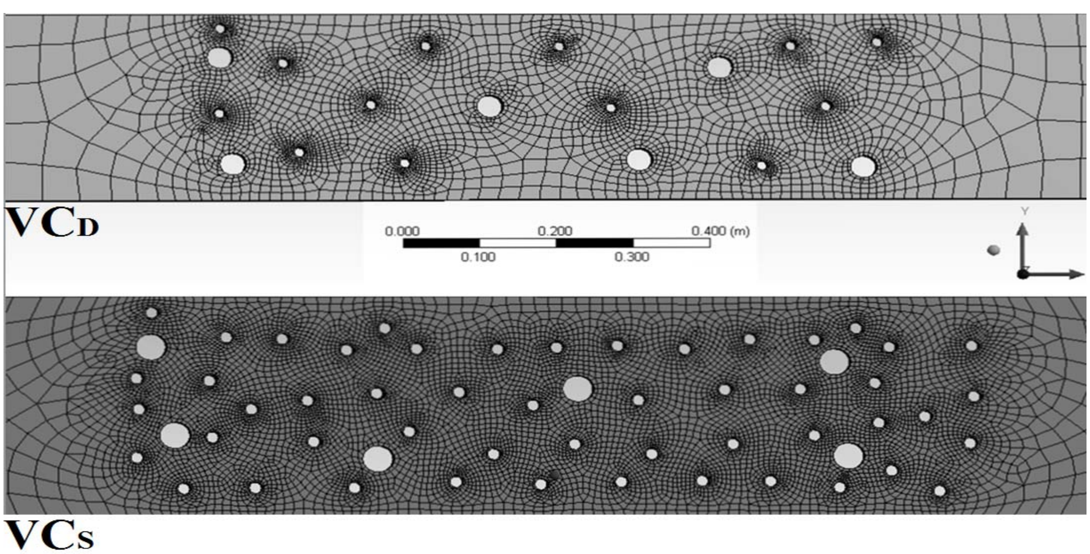

Figure-2. Plan view of the mesh for the vegetation cover (VC_D) of the deep flow region and the vegetation cover (VC_S) of the shallow flow part for the IS configurations.

ARPN Journal of Engineering and Applied Sciences
©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

Details of the experimental and CFD setup can be found in Table-1. In order to evaluate the CFD results, vertical planes (Ansys®, 2009) were created at intervals of 0.5 m along the flume and the depth-averaged horizontal velocity (U_{CFD}) (x axis) was tabulated against H (z axis). The turbulent intensity model results could not be validated so for this study the accuracy is considered dependant of the velocity similarities. Finally, the flow conditions and vegetation cover do not represent the actual flow regime of the detention pond located at Waterlooville, UK. The simulations aimed to assess only the velocity distributions within the computational domain for comparison with the experimental results. This would provide an approximation in terms of accuracy regarding the use of CFD to model emergent vegetation.

Table-1. Experimental and CFD properties for each case; Cases 1 and 2=CFD model with no vegetation, Cases 3 and 4=CFD model of the shallow flow vegetation cover replicating the actual individual stems, Cases 5 and 6= CFD model of the shallow flow vegetation with porous zone enabled, Cases 7 and 8=CFD model of the deep flow vegetation cover replicating the actual individual stems, Cases 9 and 10=CFD model of the deep flow vegetation with porous zone enabled.

	Experimental properties		**Porous zone properties**			**Mesh properties**
Case	**Q** (m^3/s)	**H at inlet** (m)	**1/α**	C_2	**ε**	**Elements**
1	0.0077	0.24	-	-	-	19200
2	0.0174	0.25	-	-	-	39000
3	0.0077	0.245	-	-	-	186225
4	0.0174	0.27	-	-	-	258020
5	0.0077	0.245	2203	13.42	0.965	39000
6	0.0174	0.27	2703	14.94	0.962	42000
7	0.0077	0.241	-	-	-	66801
8	0.0174	0.26	-	-	-	72640
9	0.0077	0.241	72.04	2.58	0.976	39000
10	0.0174	0.26	177.3	4.02	0.964	41430

3. RESULTS AND DISCUSSIONS

3.1. Experimental evaluation

Experimental results indicate that vegetation had a reducing effect on the evolving horizontal velocities upstream and downstream of the vegetation patch, as similarly observed by other researchers (Jian-tao, 2008; Ghao *et al.*, 2011; Pei-fang and Chao, 2011). Figure-3 shows U_{exp} for each of the 3 different experimental conditions under two different flow conditions. It can be seen that the more dense the vegetation, the greater the effect on flow structure, as also observed by Pei-fang and Chao (2011) and Fu-sheng (2008). In addition, the higher the discharge the greater the effect on horizontal velocity (Shucksmith, 2008). Velocity reduction is more evident for $Q_{2,}$ both upstream and downstream of the vegetation patch. The D EXP configuration presents similar velocity to the NV EXP configuration upstream and downstream of the vegetation, compared to the S EXP configuration, for both flow conditions (Q_1, Q_2). Moreover, in both flow scenarios for cases D EXP and S EXP the horizontal velocity increases within the vegetation patch (Souliotis and Prinos, 2011), possibly due to the turbulent flow patterns evolving within that region. On the other hand, the denser configuration (S EXP) presents lower horizontal velocity, both upstream and downstream of the vegetation (Pei-fang and Chao, 2011; Wen-xin *et al.*, 2012), compared to D EXP, but the velocity increase within the vegetation zone (S EXP) is more pronounced. On the basis of these findings, one could argue that the preferable vegetation density for detention ponds should not be dense in order to avoid unnecessary turbulence and velocity intensification. However, the flume experiment evaluates only the effect of the vegetation density on flow without taking into account depth variations, compared to real flow conditions at the pond. Consequently, these results are valid for VCs not differing with depth. The aim of the experimental process was to assist towards the validation of the CFD model. If the model shows similar results to the experimental findings, similar issues could be solved without the need of validation since the only variable changing would be the geometry and the vegetation density.

VOL. 8, NO. 7, JULY 2013 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences
©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

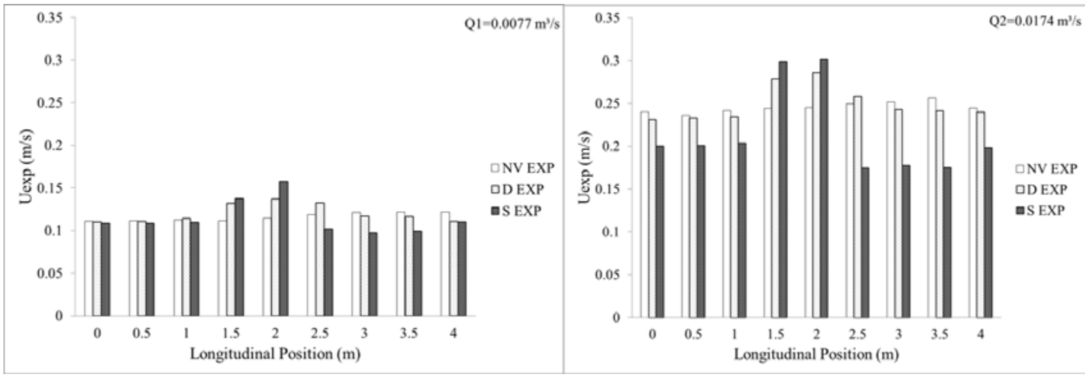

Figure-3. Experimental U of the 3 experimental configurations for each longitudinal section for Q_1 and for Q_2.

3.2. CFD evaluation

In order to evaluate the performance of the two CFD modelling strategies it is useful to compare U_{exp} at each section with U_{CFD} for each model at each section. Figure-4 shows the numerical difference (ΔU), expressed in percentage terms, between U_{exp} and U_{CFD} for each case. A negative (positive) percentage indicates numerical over- (under-) estimation compared to the model. The absolute value of ΔU (|ΔU|) for cases 1 and 2 is less than 5% for both flow conditions suggesting that the CFD model performed satisfactorily. Case 7 was also accurate with |ΔU|<5%, while for case 8 |ΔU| was slightly higher, mostly upstream and downstream of the vegetation patch. Cases 3 and 4 gave generally acceptable results, apart from at longitudinal position 3 m for case 3, where |ΔU|≈15%. Conversely, cases 9 and 10 had |ΔU| of up to 20 % within the vegetation patch, but showed good accuracy upstream and downstream of the vegetation. According to Larmaei and Mahdi (2012), the k-ε model in conjunction with the porous zone is most applicable to low density vegetated zones, also confirmed by this study. Cases 5 and 6 showed the poorest performance, in terms of prediction accuracy (|ΔU|≈30%) within the vegetation patch, producing inaccurate results. Finally, it should be highlighted that the IS configurations were much more accurate than the porous configurations in predicting turbulent flow within the vegetation. Furthermore, the CFD setup for the IS configurations had approximately |ΔU|<10%, which is less than the previously reported calculation error of 15% of Pei-fang and Chao (2011). On the other hand, the porous configurations had similar |ΔU| with a reported calculation error of 35% (Pei-fang and Chao, 2011). It is clear from these findings that the IS modelling strategy gives more accurate results than the porous zone strategy. However, the porous zone modelling approach is less time consuming than the IS approach, in terms of design effort, and could be used relatively accurately for large bodies of water with respect to the observation of general flow arrangements. Additionally, the porous zone approach could be used in an urban drainage model without the need for calibration, predicting general flow patterns relatively accurately (Larmaei and Mahdi, 2012). Furthermore, the IS strategy cannot be used to any actual pond geometry due to the difficulty of generating such a computational mesh. On the other hand, the porous zone approach can be used to any pond geometry without the requirement of generating unfeasible computational meshes. Consequently, designers could either develop micro-scale models of the actual vegetation in detention pond SuDS (typically with reeds) using the IS strategy to examine flow structures, also highlighted by Mattis *et al*. (2012), or use the porous zone approach to achieve a solution more quickly.

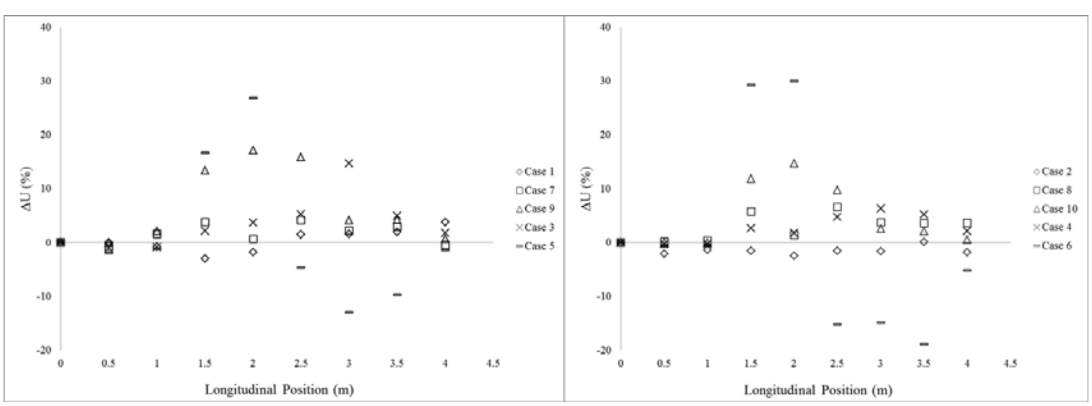

Figure-4. (a) ΔU between U_{exp} and U_{CFD} for each case versus longitudinal position in terms of the flume.

ARPN Journal of Engineering and Applied Sciences
©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

Conversely, in comparison with the use of porous zones, the IS modelling approach proved to be more effective in predicting the turbulence characteristics within the vegetated region. Figure-5 shows the turbulent intensity for cases 3, 4, 7, and 8. These results show that a less dense VC creates less turbulence upstream, both within and downstream of the vegetation. In contrast, a denser VC appears to have a much greater impact on the turbulent flow structure (0 %< I< 12.1%) upstream, within and downstream of the vegetation. For cases 3 and 4, the less turbulent flow downstream suggests that a dense VC might promote sedimentation at the downstream end of a detention pond, provided that the specific region of the pond is not vegetated. However, the less dense VC (cases 7 and 8) gives more uniformity in terms of the turbulent flow profile (0 %< I< 9%), which is preferable for detention ponds of this kind in that it might encourage sedimentation (Peterson, 1999). These observations indicate that VC has a dominant role in flow and turbulence development, as also observed by Souliotis and Prinos (2011) and Mattis *et al.* (2012). On the other hand, cases 5, 6, 7, and 8 predicted (I) rather poorly, which might explain the under-prediction of U_{CFD} within the vegetation. However, it must be highlighted that the porous zone approach performs much better if the proper User Defined Functions (UDF) are applied to the model (Ansys®, 2009). With the use of UDF, the drag and shear stress caused by a specific VC can be simulated more accurately (Ansys®, 2009). However, this approach increases design effort and time.

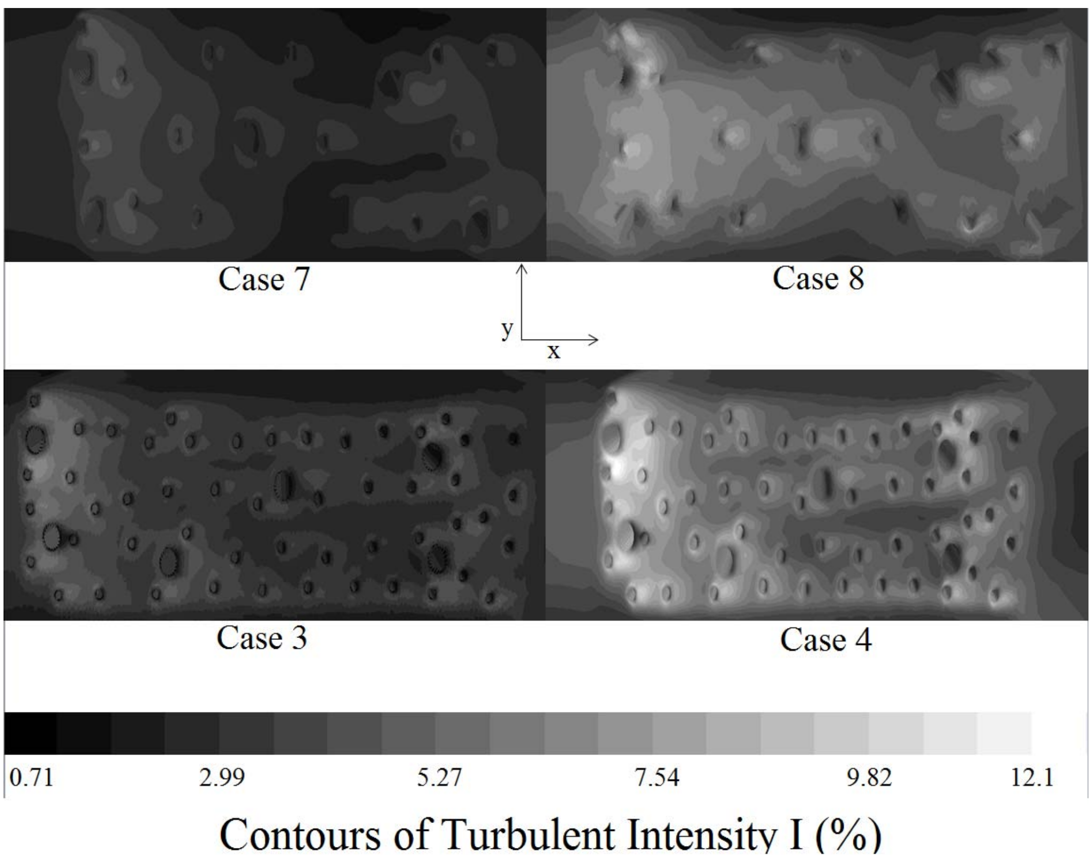

Figure-5. Plan view of contours in terms of turbulent intensity I for cases 3, 4, 7 and 8.

4. CONCLUSION

The applicability and validity of two CFD modelling strategies was evaluated, regarding the effect of emergent vegetation on open channel flows and bearing in mind time, cost, and quality, on the basis of a flume study. The first CFD model replicated the actual IS, while in the second CFD model vegetation was modelled as a porous media by use of Ansys Fluent 12.1.

Experimental results indicate that the vegetation had a significant impact on the flow structure. Increasing VC causes increasing horizontal velocity within the vegetation, while having the opposite effect on the horizontal velocity upstream and downstream the vegetation. The CFD results were in agreement for cases 1, 2, 3, 4, 7 and 8 with $|\Delta U|<10\%$. In contrast, cases 5, 6, 9 and 10 were not as accurate with $|\Delta U|<30\%$. This study confirmed that the k-ε model in conjunction with the porous zone is best applicable to low density vegetated zones and low flow conditions. Furthermore, turbulence is more pronounced in dense VC compared to sparse VC. Consequently, a dense VC may reduce the horizontal flow velocity upstream and downstream, but the increased

ARPN Journal of Engineering and Applied Sciences
©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

turbulence might cause re-suspension of deposited sediment within the vegetated region. Furthermore, a more sophisticated approach (UDF) of the porous zone strategy might result in better prediction capability of Ansys Fluent; but with increased time requirements in terms of the design process.

The findings of this study suggest that it is possible to examine the flow characteristics of a given vegetation profile without the need to develop unfeasible and impractical numerical models by using an appropriate CFD modelling strategy. This scheme involves the employment of an accurate, cost-effective, and less time-consuming micro-scale/porous media modelling approach to evaluate the flow structure in a vegetated detention pond SuDS. This paper is of vital importance to civil engineers that design pond systems and are interested in assessing, at pre-construction stage, the effect of emergent reeds vegetation on the evolving flow patterns without consuming a great deal of time while generating hydraulically efficient proposals.

REFERENCES

ANSYS® Academic Research. 2009. Release 12.1. Fluent Documentation. ANSYS Inc. Chapters 1-5.

Choi S. and Kang H. 2006. Numerical investigations of mean flow and turbulent structures of partly-vegetated open-channel flows using the Reynolds stress model. Journal of Hydraulic Research. 44(3): 203-217.

Folkard MA. 2011. Vegetated flows in their environmental context: a review. Proceedings of the Institution of Civil Engineers: Engineering and Computational Mechanics. 164(1): 3-24.

Gao G., Falconer RA. and Lin B. 2011. Modelling open channel flows with vegetation using a three-dimensional model. Journal of Water Resource Protection. 3: 114-119.

Green JC. 2005. Modelling channel resistance in vegetated streams: review and development of new theory. Hydrological Processes. 19(6): 1245-1249.

Green JC. 2006. Effect of macrophyte spatial variability on channel resistance. Advances in Water Resources. 29(3): 426-438.

Er-qing H. and Xing-e H. 2010. A study of drag coefficient related with vegetation based on the flume experiment. Journal of Hydrodynamics. 22(3): 329-337.

Feng-feng G. 2007. Roughness coefficient for unsubmerged and submerged reed. Journal of Hydrodynamics. 19(4): 421-428.

Fu-sheng W. 2008. Characteristics of flow resistance in open channels with non-submerged rigid vegetation. Journal of Hydrodynamics. 20(2): 239-245.

Hamill L. 2001. Understanding hydraulics (2nd ed.). New York : Palgrave Macmillan.

Järvelä J. 2002. Flow resistance of flexible and stiff vegetation: a flume study with natural plants. Journal of Hydrology. 269: 44-54.

Jian-tao Z. 2008. Numerical model for flow motion with vegetation. Journal of Hydrodynamics. 20(2): 172-178.

Jordanova AA., James CS. and Birkhead AL. 2006. Practical estimation of flow resistance through emergent vegetation. Proceedings of the Institution of Civil Engineers: Water Management. 159(3): 173-181.

Larmaei MM and Mahdi TF. 2012. Depth-averaged turbulent heat and fluid flow in a vegetated porous medium. International Journal of Heat Mass Transfer. 55: 848-863.

Li CW. and Zeng C. 2009. 3D numerical modelling of flow divisions at open channel junctions with or without vegetation. Advances in Water Resources. 32: 49-60.

Liwei F., Reti H., Wenxing W., Zexiang L. and Zhiming Y. 2008. Application of computational fluid dynamic to model the hydraulic performance of subsurface flow wetlands. Journal of Environmental Science. 20: 1415-1422.

Mattis SA., Dawson CN., Kees CE. and Farthing MW. 2012. Numerical modelling of drag for flow through vegetated domains and porous structures. Advances in Water Resources. 39: 44-59.

Minitab Inc. 2009. Statistical analysis software, version 6.2.2, Minitab Ltd, U.K.

Naden P., Rameshwaran P., Mountford O. and Robertson C. 2006. The influence of macrophyte growth, typical of eutrophic conditions, on river flow velocities and turbulence production. Hydrological Processes. 20(18): 3915-3938.

Pei-fang W. and Chao W. 2011. Numerical model for flow through submerged vegetation regions in a shallow lake. Journal of Hydrodynamics. 23: 170-178.

Peterson EL. 1999. Observations of pond hydrodynamics. Journal of Agricultural Engineering. 21(2000): 246-269.

Saggiori S. 2010. CFD modelling of solute transport in vegetated flow. M.Sc Thesis, University of Sheffield, U.K.

Shih TH., Liou WW., Shabbir A., Yang Z. and Zhu J. 1995. A new **k-**□ eddy viscosity model for high reynolds number turbulent flows. Computers and Fluids. 24(3): 227-238.

©2006-2013 Asian Research Publishing Network (ARPN). All rights reserved.

Shucksmith D. J. 2008. Impact of vegetation in open channels on flow resistance and solute mixing. PhD Thesis. University of Sheffield, U.K.

Souliotis D. and Prinos P. 2011. Effect of a vegetation patch on turbulent channel flow. Journal of Hydraulic Research. 49(2): 157-167.

Stovin V., Guymer I., Lacoursiere O. J. and Vought B-M. L. 2009. The effects of vegetation on residence times in ponds. Paper presented at the SUDSNet National Conference, November 12-13. Retrieved 21/06/2012 from: http://sudsnet.abertay.ac.uk/.

Wen-xin H., Cheng-quang L., Yu-hong Z., Zhong-dong Q. and Zhong-hua Y. 2012. Curved open channel flow on vegetation roughened inner bank. Journal of Hydrodynamics. 24(1): 124-129.

Yan Y. 2011. Development of a coupled CFD system code capability (with a modified porous media model) and its applications to simulate current and next generation reactors. PhD Thesis. University of Illinois, USA.