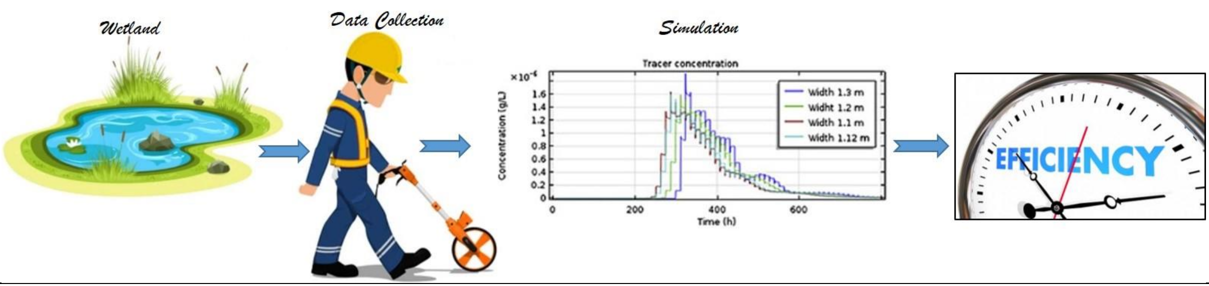
Journal of Applied Research in Water and Wastewater
E-ISSN: 2476-6283
Journal homepage: https://arww.razi.ac.ir

Hydraulic performance of constructed wetland at NUST H-12 campus

Laraib Pervaiz Akhter[*,1,2], Hamza Gabriel[1,2], Sajjad Haider[1,2], Shatirah Akib[3]

[1]Department of Water Resources Engineering and Management, School of Civil and Environmental Engineering (SCEE), National University of Sciences & Technology (NUST), Islamabad, Pakistan.
[2]NUST Institute of Civil Engineering (NICE), National University of Sciences & Technology (NUST), Islamabad, Pakistan.
[3]School of Architecture, Design and the Built Environment, Nottingham Trent University, Nottingham, UK.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:
Received 10 September 2021
Reviewed 15 November 2021
Received in revised form 23 December 2021
Accepted 26 December 2021
Available online 29 December 2021

Keywords:
Constructed wetland
COMSOL
Computational fluid dynamics
Simulation modelling
Turbulent model
Water reuse

Article type: Research Article

ABSTRACT

The paper narrates a study of a numerical model taken into consideration to explore the overall hydraulic performance of Constructed wetland located at NUST H-12 Campus, Islamabad. The governing equations of flow in wetland ponds and the transport particle were solved using COMSOL Multiphysics 5.0. In this study, the simulation of the model is done to find the dead spaces and short-circuiting in the wetland along with providing remedial measures to reduce the dead spaces. The comparison of two turbulent models i.e. K-ω and K-ε were also made to depict the velocity of wetland and the particle tracer study was also conducted to find the behavior of wetland. The model was simulated with experimental data and the results revealed that 15% to 20% area of wetland is experiencing short-circuiting. Alternative wetland designs were suggested for the same flow condition. The K-ω model was considered to be more suitable due to the limitation of K-ε model.

1. Introduction

Majority of urban cities in Pakistan is facing a huge crisis because of an increase in population growth which has posed a serious threat in the provision of infrastructure services. Out of many services, wastewater management is one of the foremost infrastructure services whose demand has exceeded the overall supply of the country. The use of septic tanks is considered as the prevalent method for disposal of households and other on-site wastewater treatment; however, it has been failed to achieve the quality standards for direct disposal into the water bodies. To confront, the construction of wetlands should be done in those urban areas. Constructed wetlands (CW) are the low-cost sanitation system principally designed for the primary treatment of wastewater (Eslamian et al. 2020). They are increasingly being used to handle different types of wastewaters like domestic wastewater (Chang et al. 2012); landfill leachate (Johnson et al. 1999) and diary effluent (Tanner et al. 2005). CW is often used for secondary treatment of municipal wastewater. In wetlands, the patterns of water movement are the controlling factor for the wastewater treatment. The poor hydrodynamic behavior was observed for horizontal subsurface flow wetland because of porosity, sedimentation, root growth, and adsorption. The biofilm development creates a variation in preferential flow paths during the operating time. The biofilm also induces variations in hydraulic residence time distribution and consequently the efficiency of treatment. (Persson et al. 1999) stated, "The flow pattern and flow velocities of CW in COMSOL Multiphysics cannot provide insight about water quality issues unless a tracer study is carried out." Many studies have evaluated the hydrodynamics of CW and majority concludes that conventional designs of the system were favorable for developing preferential flow paths. According to Persson and Peterson "Efficient Pond geometry can help to reduce horizontal velocity gradients by encouraging a more uniform flow profile and minimizing the amount of recirculation." The k–ε model is a basic two-equation transport model used in computational fluid dynamics (CFD) where k referred to the turbulence kinetic energy, first transported variable and ε referred to the rate of dissipation of turbulence energy (Hinze. 1975). The K-ω is also

*Corresponding author Email: engr.laraib93@gmail.com

How to cite: L. Pervaiz Akhter, H. Gabriel, S. Haider, S. Akib, Hydraulic performance of constructed wetland at NUST H-12 campus, *Journal of Applied Research in Water and Wastewater*, 8 (2), 20121, 169-173.

a PDE equation a first order turbulence model used in CFD primarily used for the near-wall treatment, where k referred to turbulence kinetic energy and ω referred to the specific dissipation rate (Wilcox. 1988). On the other hand, as it was highlighted by Liwei et al. (2008), the hydrodynamic evaluation of CW using physical tracer experiments are so expensive, time-consuming and sometimes impractical. Henceforth, the flow patterns in the wetland along with the suitability of multiple geometries can be better understand by using the numerical model as a design tool. It also helps in facilitating the development of optimal design with respect to the characteristics of flow.

From the literature review it was observed that internationally, great efforts were made to predict the overall efficiency of CW by using 2D and 3D models (Ta and Brignal.1998; Wood et al. 1995; Rengers et al. 2016; Guo et al. 2020; Wang et al. 2021 a, b). However, in Pakistan, the scope of finding the hydraulic performance of Wetland by using the COMSOL model is nearly zero. According to the review of Mortaza et al. (2010), there exists very limited information on the bioremediation of effluents and soils physical treatment of effluents and soils, thermal remediation strategies, waste landfills, chemical oxidation treatments and amendments to decrease bioavailability. The restrictions present in this study solely depends upon its high implementation cost. Up till now, only one research is made at the national level on the said topic. This research provides an extension of earlier work conducted in 2018 who successfully run the model on hypothetical values. In this research, the on-site data was collected to be used as an input for further simulation by using COMSOL Multiphysics 5.0.

COMSOL Multiphysics is a simulation software used in major study fields like engineering, manufacturing and in scientific research through modeling design and devices. It computes a multiphysics problem simultaneously instead of one type of physics. The predefined interface of COMSOL combines boundary conditions, physical limitations, loads fluxes, material properties and other physical quantities. All these quantities were combined with the geometry of the model, the related equations were used and the study is computed to generate the results (COMSOL Multiphysics Reference Manual. 2019).

The aim of this study was to investigate the hydraulic efficiency of the constructed wetland at NUST H-12 CAMPUS in Islamabad by means of Computational Fluid Dynamics (CFD) approach. Compared to the global approach commonly used to analyze performance of wastewater treatment process involved in a given constructed wetland, the CFD approach allows to reproduce adequately the 3D structure of mean and turbulent flows in each pond constituting the CW, which implies a more realistic reproducing of several involved transport mechanisms in this wastewater treatment process. Therefore, this approach should be useful to study the eventual existing of dead zones as well as short-circuiting in the considered CW.

This work is structured in three sections. After an introducing section, we attempt at the second one to describe the adopted approach as well as its numerical set up. The numerical results are analyzed and interpreted in the third section before synthesizing the main conclusions at the last section.

1. Method

2.1. Constructed wetland at NUST

The CW is located at NUST H-12 Campus Islamabad. This city, located at an elevation equal 540 meters, has a humid subtropical climate where the greatest amount of precipitation occurs in July with an average of 174 mm. while the driest month is November with amount of precipitation around 30 mm. On the other hand, the monthly averaged temperature usually reaches its maximum in June with mean value around 30 °C. The total area of CW is around 1000 acres situated in a public residential sector established in 1991. The CW accumulates waste of around 15 schools and institutes including hostels and other major amenities, covering a total population of around 6000. The geographical coordinate of CW is 33°38'31.1" N and 73°00'13.7" E. It comprises of eight ponds which are connected in series as shown in Fig.1. After the eighth pond, there is a FILTER plot irrigated with typha latifola.

The waste accumulated from all the residential and institutional areas towards the sedimentation tank through a V notch, from there it is directed to the eight ponds. In order to avoid the contamination of underlying aquifer ponds, the lining of High-Density Polyethylene is used. In the last pond, aeration is done by using solar energy. All the ponds are lined with low-density polyethylene to avoid contamination of the underlying aquifer. The effluent from CW discharge to FILTER plot for further improvement in the quality of wastewater. The whole system of CW is based on gravity flow.

Fig. 1. Constructed wetlands at NUST.

All eight ponds are symmetrical, each having a dimension of 12.95m X 6.85m X 2.13m. The on-site data of stream flow velocities were conducted. According to the experimental deducing of Daniel et al. (2008), the velocity flow meter is used to measure and record velocities of each pond at a depth of 0.2D, 0.6D and 0.8D where D is the effective depth of flow.

The flow of water within a CW is the major contributing factor in the treatment of biological, chemical and physical processes. Therefore, the hydraulic performance has a significant impact on the efficiency of CW. As noted by Jenkins et al. (2005) "the hydrodynamic characteristics within a CW system are affected by features such as:

- Mixing
- Wetland Geometry
- Wetland Bathymetry
- Hydraulic characteristic of inlet and outlet structures
- Vegetation type

The longer the wastewater retained in the pond, the higher the probability of removal of pathogens. The time required for water to stay in the wetland is called Hydraulic Retention Time (HRT). It is defined as the volume of water per total flow rate capacity. Equation (1) indicates the formula for calculating the HRT.

$$HRT = \frac{V}{Q} \tag{1}$$

where, V stands for the volume of each pond in cubic meter and Q is defined as the total flow rate capacity in cubic meters per second. The total daily discharge handled by the CW at full capacity is 0.00525 m^3/sec.

2.2 COMSOL Model:

COMSOL is the physics simulation software where Partial Differential Equations (PDEs) and Finite Element Method (FEM) are solved. It has various convenient features that made this software beneficial to the many engineers. COMSOL software is used for the simulation and modeling of real-world Multiphysics. Consequently, it has become a leading computing software. The software includes various applications such as fluid, electrical, mechanical chemical, etc. It has provided an integrated simulation platform to the present day's researchers and engineers to design the model in a short time interval. The basic numerical theory used in COMSOL model is governed by the Navier Stroke equations:

$$\rho \frac{\partial u}{\partial t} - \nabla . \left[\eta \left(\nabla u + \left(\nabla u \right)^T \right) \right] + \nabla p = F \tag{2}$$

$$\nabla . u = 0 \tag{3}$$

where η stands for dynamic viscosity measured in $ML^{-1}T^{-1}$, u stands for velocity vector measured in LT^{-1}, ρ referred to the density of fluid measured in ML^{-3}, p is the pressure measured in $ML^{-1}T^{2}$, F referred to the body force measured in $ML^{-2}T^{-2}$ and ∇ is a mathematical operator.

The CW is lined with impermeable membrane, therefore, the boundary condition for the velocity is no-penetration condition u.n=0

$$u_T = \frac{|u|}{\frac{1}{K_v} Ln(\delta_w^*) + B} \tag{4}$$

where, u, initial inlet velocity which is 0.7m/s Kv, von Kárman constant with a value of 0.41, B is the roughness factor, for turbulent flow the value of B is 8.5. In comparison with K-ω turbulent model, K-ε model fails to perform in transitional flow and flow with an adverse pressure gradient. The schematic diagram of CW is shown in Fig. 2. indicating the inlet and outlet positions.

First of all, software for the analysis of COMSOL Multiphysics 5.0 is opened, and the selection of the turbulent model is done for the simulation. After the flow model selection, the study type was selected as stationary where field variables does not change with respect to time.

Inlet: In order to find the hydraulic performance of CW, the velocity data was collected on at every pond at various depths. The frequency of data collection was a daily basis for three consecutive months. The procedure used to collect data involves averaging stream flow velocities at 0.2D, 0.6D and 0.8D, where D is the effective depth of flow. The data was obtained in frequency Hz form with the help of a velocity probe from each CW. The maximum average velocity from Pond 1 was selected as an inlet velocity which is 0.7 m/s.

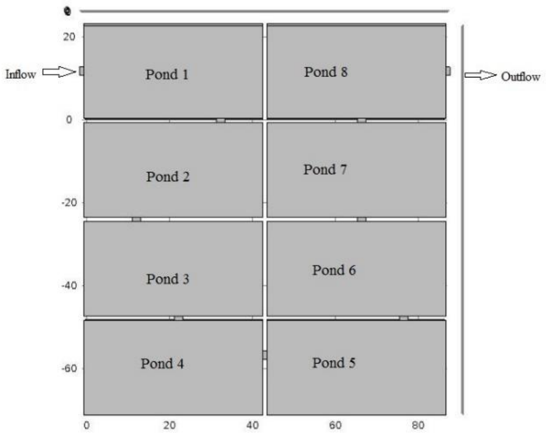


Fig. 2. Schematic diagram of CW.

Outlet: The pressure was selected zero at the boundary condition.

Meshing: Meshing is yet another important step for the simulation of turbulent flow in COMSOL under fluid analysis. The Mesh is used to divide the geometry of the model into small elements, referred to as the mesh element. It divides the geometry into various shapes like triangular, hexagonal, pyramid, etc. The mesh size ranges from extremely fine to extremely coarse. The mesh size is directly proportional to the time required for computation. Smaller mesh size results in more mesh element hence more time is required for computation and vice versa. The coarser element size is selected for this particular study. Mesh size is directly proportional to the time required for computation. Smaller mesh size results in more mesh element hence more time is required for computation and vice versa. The coarser element size is selected for this particular study and the model is build as per desired meshing. It has been opted in this study to conceive and mesh a numerical domain corresponding to the whole of wetland even the eight are symmetrical. If computing capacities allow it, such choice should be the most appropriate as it is the most representative of the real physical domain. The maximum element size is taken as 0.98m and the minimum element is 0.304m. A total of 77000 cell elements are used in mesh geometry. The x, y and z-direction was set as 1 cell element.

For analyzing the tracer transport in COMSOL 5.0 model, the transport of diluted species features is used. It helps in computing the concentration field of a dilute solute in a solvent. The basic phenomenon of tracer diffusion and convection are modeled by the mass conservation equation. The non-conservative formulation of the convective term given as:

$$\frac{\partial c}{\partial t} + u.\nabla c = \nabla.\left(D\nabla c\right) + R \quad (5)$$

where, c referred to the concentration of species measured in mol/ m3, D stands for the diffusion constant in m^2/sec and R stands for reaction rate expression for the species measured in mol/(m^3s). c is referred to the concentration of species measured in mol/m^3, D stands for the diffusion constant in m^2/sec and R stands for reaction rate expression for the species measured in mol/(m^3s)

The initial concentration of tracer at the inlet of CW is assumed to be zero while at inflow the initial concentration is given as 1 mol/ m^3. The diffusion coefficient is a scalar value given as 1e-9[m2/s] with water as a material. The mass flow through the outlet boundary was specified as convective dominated (-n.Di ∇ci= 0). It assumes that mass flux because of diffusion across this boundary is zero. The insulation condition is applied at the boundaries assuming that no mass is transported across them. The dynamic viscosity "η" provided was 0.001 Pa.s and the density of water "ρ" was 1000 kg/m^3.

As the adopted CFD approach in this work fits in the framework of the Reynolds averaged numerical simulation (RANS) approach, it will be assumed that the flow turbulence should be fully developed such that the local isotropy of the turbulent microstructures is verified according to the Kolmogorov theory (Shiestel. 2008). We note equally that the flow in the wetland is assumed to be adiabatic. This implies that there is no heat transfer with the surrounding environment. As well, the surrounding ambient temperature has no influence on the deduced numerical results.

3. Results and discussions

3.1. HRT

From the previous studies it was found that the daily capacity of CW treatment facility is 0.00328 m^3/s, water depth was calculated on daily basis for three consecutive months for every pond and the average depth is taken for each pond. Table 1 shows the calculation of HRT resulting in 3.2 days which is an effective HRT as reported in Metcalf and Eddy (1991).

Table 1. Calculation of HRT.

Description	Length, m	Breadth, m	Depth of water, m	Total capacity		Hydraulic retention time (HRT)	
				m^3	US gallon	days	h
Pond 1	12.96	6.86	0.91	81.30	21458	0.29	6.87
Pond 2	12.96	6.86	1.37	121.94	32187	0.43	10.30
Pond 3	12.96	6.8	1.22	108.4	28611	0.38	9.16
Pond 4	12.96	6.86	1.52	135.49	35764	0.48	11.44
Pond 5	12.96	6.86	1.98	176.14	46493	0.62	14.88
Pond 6	12.96	6.86	1.34	119.23	31472	0.42	10.07
Pond 7	12.96	6.86	1.22	108.40	28611	0.38	9.16
Pond 8	12.96	6.86	0.75	66.39	17524	0.23	5.61
			Total	**917.3**	**242121**	**3.23**	**77.48**

3.2. Simulation of existing design of CW

In Fig. 3, it has observed that velocity distribution depends upon the configuration of the inlet and outlet pipe. In pond 1 the flow enters from the midsection of the pond and spread evenly throughout; no dead zone was observed. In pond 2, the inlet and outlet pipes are located at a distance of 3.65m from their nearest corner, the alignment of pipes is such that a small dead zone was observed. In pond 3 relatively large dead zone was observed as compared to pond 2; however, in pond 4 the configuration of inlet and outlet pipes are change because of the geometrical constraint of the pond. This caused the occurrence of a large dead zone which results in effecting the treatment of wastewater. In pond 5 and 6, the pipes were configured in a diagonal direction to each other therefore low dead zone observed in the pond as water passing from inlet have easy access to flow all across the pond before reaching the outlet. In pond 7 the placement of inlet and outlet pipes was such that almost large area of the pond is under no velocity zone and zero recirculation observed. In the last pond, the inlet and outlet pipes were placed at the farthest points from each other thereby making it easy for the water to flow all across the pond. Based on empirical analyses and visual performance of the CW, 15% ~ 20% of the area falls under the dead zone.

3.3. Simulation of modification in existing design of CW

In order to minimize the effect of short-circuiting and dead zone, the modification in the design of CW was made. After simulation, it was observed that if pipes of pond 4 and pond 7 are placed at the center then the flow is distributed evenly throughout the pond. From the modifications made in the pond, it was observed that the overall distribution of wastewater in the treatment facility has improved. Fig.4 shows the velocity distribution of ponds after modifications in CW.

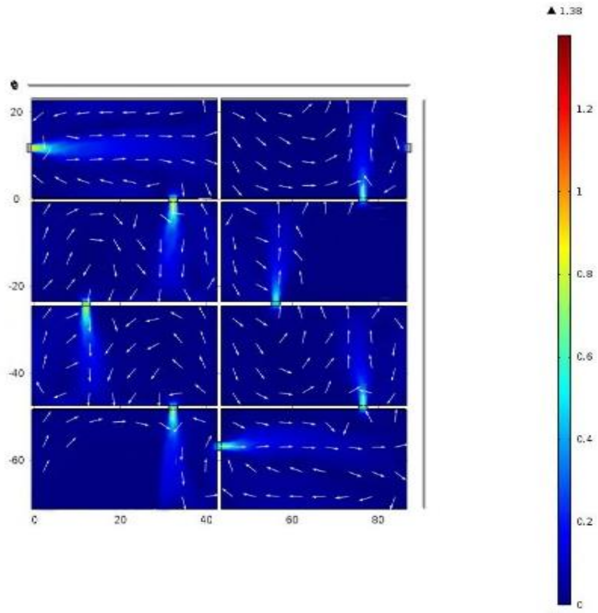


Fig. 1. Velocity distribution in existing CW.

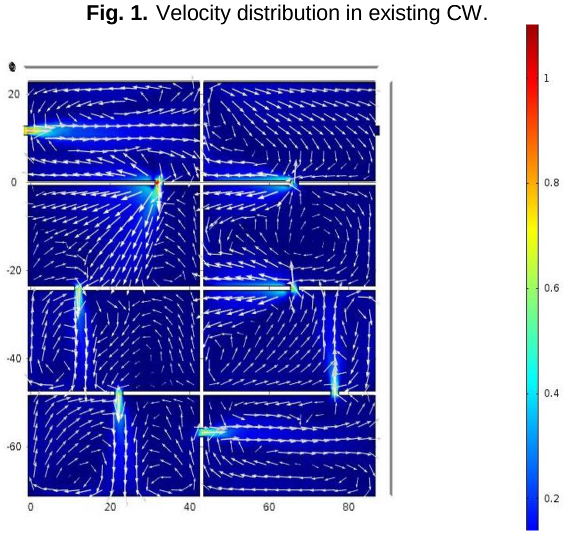


Fig. 2. Velocity obtained after modifications of CW.

3.4. Comparison between K-ε and K-ω on existing CW

The wastewater treatment facility of CW is modeled by using two turbulent models i.e. K-ε and K-ω and then comparing their results. The flow type is turbulent so rough wall function is applied as a boundary condition. No attempts were made to find the pressure at inlet and outlet as losses due to forces such as external forces and shear forces are neglected. The velocity is a dominant factor in simulation and the majority of the analytical results were based on velocity head from conservation of mass. The graphical representation obtained from both K-ε and K-ω model is shown in Fig.5. The major difference in results obtained from K-ω and K-ε model is probably because of incorporation of low Reynold number modification. The K-ω model has the capability to forecast wall-bounded flow with good precision. Pope in 2000 and Devaud in 2011 stated that the K-ε model has the ability to calculate the flat boundary condition while lacks in sudden contractions and far wake obstacles, on the contrary, the K-ω model undertake the treatment of near-wall regions while fails to predict the treatment of non-turbulent free stream boundaries.

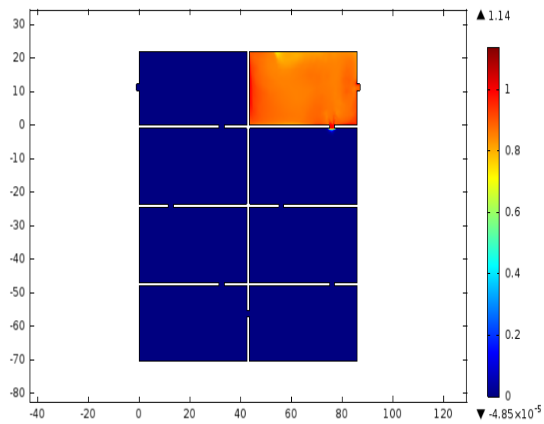


Fig. 3. Particle tracing of CW at pond 8.

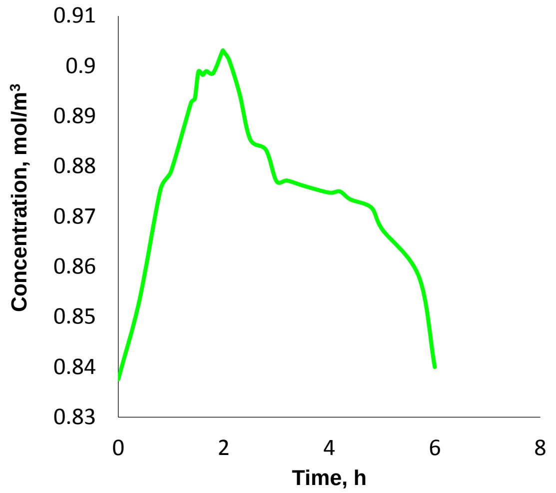


Fig. 4. Graphical representation of concentration of particle tracing at pond 8.

4. Conclusions and recommendations

After running the model successfully, the output obtained from the model needs to be scrutinized that whether or not the results obtained from simulation make physical sense. Based on empirical analyses and visual performance of the CW, 15%~20% of the area falls under the dead zone. Thus, the overall efficiency of CW is found to be approximately 75 %~80 %. The modifications made in CW have significant influence in removing the dead zones of the pond and increasing the overall hydraulic performance of CW which consequently shows a significant impact on the efficiency of the wastewater purification process. The comparison between two turbulent models was made and despite the fundamental difference in the nature of the turbulence model, the difference found was small less than 10 %. The K-ε model lacks the ability in sudden contraction and in rotating flow simulation therefore, K-ω model was considered more suitable for this scenario. The tracer concentration curve depicts the theoretical correct simulation of the CW. The model results obtained after simulation can be validated by using field data. To validate the CFD model used in the simulation, concentration data need to be obtained experimentally and compared to those predicted by the model. For future study, pond 8 needs to be selected due to the fact that pond is not covered with vegetation and hence field observation will be made easier and the values obtained from the model will be used for validation. COMSOL is considered as an effective tool for evaluating flow behavior in CW. It is used during the research or design phase of CW hence reducing the overall operating and maintenance cost. At the end, comparing to the global modeling approach commonly used for the secondary treatment of wastewater, the local modeling approach adopted in this work seems to be so wise even it requires more computing capacities than the global one. In fact, such adopted approach using CFD simulation allows to reproduce locally all the mean as well as the turbulent characteristics of the flow through the whole flow domain. Therefore, the numerical results given by this approach should be so helpful to analyze for example the eventual existence of dead zones or short-circuiting in the constructed wetland, as well as, to test new suggested variants in order to avoid them. Nevertheless, the presented study in this work, is limited to the investigation of the hydrodynamics of the wetland. Which seems to be still insufficient to achieve a complete study of the wastewater secondary treatment efficiency. Indeed, such target requires to take into consideration the biological activities involved in the wetland. This needs a modeling of the activated sludge as well its coupling with the developed hydrodynamics modeling in this work. This may be considered as a so interesting perspective in the future of this work.

Nomenclature

CFD	Computational fluid dynamics
CW	Constructed wetland
FEM	Finite element method
PDEs	Partial differential equations
RANS	Reynolds averaged numerical simulation
HRT	Hydraulic retention time

References

Barr D., Coefficient of permeability determined by measurable parameters, Ground Water 39 (2000) 356-361.

Chang J., Wu S., Dai Y., Lianga W., Wu Z., Treatment performance of integrated vertical-flow constructed wetland plots for domestic wastewater, Ecological Engineering 44 (2012) 152-159.

Eslamian S., Okhravi S., Eslamian F., Constructed wetland, Hydraulic design, 1st Ed., Taylor and Francis, CRC Press, USA; (2020).

Guinot V., and Soares-Farzao S., Flux and source term discretization in two-dimensional shallow water models with porosity on unstructured grids, International Journal for Numerical Methods in Fluids 50 (2006) 309-345.

Guo C., Yuanlai Cui Y., Luo Y., Response of solute transport model parameters to the combination of multiple design parameters and their quantitative expression with hydraulic indicators of FWS-constructed wetlands, Environmental Science and Pollution Research 27 (2020) 43283-43295.

Hinze J.O., Turbulence, Second Edition., McGraw-Hill, New York, (1975).

Jenkins G.A., Greenway M., The hydraulic efficiency of fringing vegetation in constructed wetlands, Ecological engineering 25 (2005) 61–72.

Johnson K.D., Martin C.D., Moshiri G.A., Mc. Crory W.C., Performance of a constructed wetland leachate treatment system at the Chunchula landfill, Mobile Country Albama, Chapter 5, CRC Press/Lewis Publishers: Boca Raton, FL. USA (1999) p. 57-70.

Liwei F., Reti H., Wenxing W., Zexiang L., Zhiming Y., Application of computational fluid dynamic to model the hydraulic performance of subsurface flow wetland, Journal of Environmental Sciences 20 (2008) 1415-1422.

Metcalf E., and Eddy H.P., Wastewater engineering: treatment disposal and reuse, McGraw Hill, New York, (1991).

Murtaza G., Ghafoor A., Qadir M., Owens G., Aziz M.A., Zia M.H., Saifullah, Disposal and use of sewage on agricultural lands in Pakistan: A review, Pedosphere 20 (2010) 23-34.

Persson J., Somes N.L.G., Wong T.H.F., Hydraulic efficiency of constructed wetlands and ponds, Water Science and Technology 40 (1999) 291-300.

Rengers E.E, Barbosa da S., Paulo P.L., Hydraulic performance of a modified constructed wetland system through a CFD-based approach, Journal of Hydro-environment Research 12 (2016) 91-104.

Schiestel R., Modeling and simulation of turbulent flows, 1st Ed. ISTE USA, (2008).

Ta C.T., and Brignal W.J., Application of computational fluid dynamics technique to storage reservoir studies, Water Science and Technology 37 (1998) 219–226.

Tanner C.C., Nguyen M.L., Sukias J.P.S., Nutrient removal by a constructed wetland treating subsurface drainage from a grazed dairy pasture, Agriculture Ecosystems and Environment 105 (2005) 145-162.

Thackston E.L., Jr F.D.S., Schroeder P.R., Residence time distribution of shallow basins, Journal of Environmental Engineering 113 (1987) 1319-1332.

Wang R., Xu L., Xu X., Xu Z., Cong X., Kong K., Hydraulic characteristics of small-scale constructed wetland based on residence time distribution, Environmental Technology (2021a) 1-10.

Wang R., Xu L., Xu X., Xu Z., Cong X., Simulation and optimization of hydraulic performance of small baffled subsurface flow constructed wetland, Water Science and Technology 84 (2021b) 632-643.

Wilcox D.C., Reassessment of the scale-determination equation for advanced turbulence models, AIAA Journal 26 (1988) 1299-1310.

Wood M.G., Greenfield P.F., Howes T., Johns M.R., Keller J., Computational fluid dynamic modeling of wastewater ponds to improve design, Water Science and Technology 31 (1995) 111–118.