Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 186 (2017) 84 – 92

Procedia Engineering

www.elsevier.com/locate/procedia

XVIII International Conference on Water Distribution Systems Analysis, WDSA2016

Characterization of modular deposits for urban drainage networks using CFD techniques

Sánchez-Beltrán, H.[b]; Montes Rodríguez, C.D.[a]; Barrera Triviño, J.C.[a]; Iglesias-Rey, P.L.[b]; Saldarriaga Valderrama, J. *; Martínez-Solano, F.J.[b]

[a]*CIACUA (Centro de Investigaciones en Acueducto y Alcantarillados). Universidad de Los Andes (Colombia).*
email: cd.montes1256@uniandes.edu.co; jc.barrera147@uniandes.edu.co; jsaldarr@uniandes.edu.co
[b]*Dpto. Ingeniería Hidráulica y Medio Ambiente. Universidad Politécnica de Valencia (España),*
email: hecsanbe@cam.upv.es; piglesia@upv.es; jmsolano@upv.es

Abstract

Storm tanks are an increasingly solution used to control floods caused by the increase of runoff flows generated by rain. To expedite the construction of these infrastructures and reduce downtime of urban supplies plastic modular elements as basic building structures are used. This paper presents the characterization of this sort of modular elements determining its energy dissipation to the passage of the water and comparing the hydraulic behavior of modular deposits to conventional ones.

This paper justifies the use of computational fluid dynamic techniques (CFD) for characterizing hydraulic behavior of these modular elements based on three-dimensional models. These models have been validated by an experimental prototype built in laboratory. This one consists of a channel along which various completely monitored modular blocks are disposed in order to analyze both, circulating flows and water levels reached at different points of the system. The final result is a validated and calibrated model that allows to represent the behavior of this kind of infrastructures against strong intensity storms.

A one-dimensional model of such structures is proposed as a way to join it with a specific computational model of drainage networks analysis as is the case of SWMM. Finally, a comparison between the results obtained by CFD techniques and those obtained after adjustment in SWMM is shown.

---

* Corresponding author .Tel.: +571-339-4949 Ext. 2805.
E-mail address: jsaldarr@uniandes.edu.co

Peer-review under responsibility of the organizing committee of the XVIII International Conference on Water Distribution Systems
doi:10.1016/j.proeng.2017.03.212

Peer-review under responsibility of the organizing committee of the XVIII International Conference on Water Distribution Systems

Keywords: CFD; sustainable urban drainage systems; holding tanks; modular tanks.

1. Introduction

The growing urban development of population centers in much of the world joined with the significant effects of climate change are causing an increasingly important and recurring increase of the damage caused by flooding. Much of the drainage networks of cities were designed for precipitation characteristics and return periods that have proved to be insufficient with the lapse of time. Therefore, solutions need to be addressed both to reduce runoff generated flows as to control circulating ones through the rainwater drainage networks.

All these flow control rain technologies are commonly known as SUDS (Sustainable Urban Drainage), term that encompasses a multitude of solutions to control runoff although many of them require significant costs that make them practically unviable. Therefore, not only should focus on reducing runoff input to the network but also in the flow control techniques development. The idea is to design strategies to reduce flow rain peaks and maximize the capacity of existing networks.

The use of detention and storm tanks for flood control is a solution increasingly used as an alternative one to control increased rainfall caused by climate change [1].

Nature and execution of storm tanks can be very diverse, from conventional way based on concrete structures to the most innovative ones in which modular structures are employed to improve the construction speed if many modular units are required at the same time that minimizing urban supply disruption is achieved.

Currently, a wide range of modular structures exists on the market with both, different geometries and sizes. In this study the Aquacell brand supplied by Mexichem-PAVCO in Colombia shown in Fig. 1 has been chosen for the development of this study.

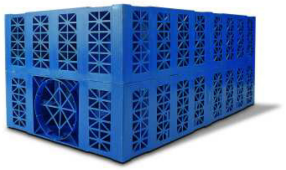

Fig. 1. Modular Aquacell® element for the construction of holding tanks.

Flow developed through these modular structures is much more complex than present in conventional deposits creating greater energy dissipation inside. This energy dissipation has two effects on the overall tank performance. On the one hand, energy loss between adjacent modules makes the tank not be filled with a horizontal water sheet so the depth in the tank inlet may be substantially higher than the draft at either of its ends. Moreover, energy dissipation also implies a flow deceleration, phenomenon that under certain drainage network circumstances may contribute to buffer the flow and mitigate possible flooding problems.

2. Objectives

The purpose of this paper is to analyse the behaviour of this sort of modular structures. Particularly, the behaviour of a specific item will be discussed: the module called Aquacell from Wavin company and provided by Mechichem-PAVCO Colombia company for this study. The aim pursued is to better understand the behaviour of these modules in order to extrapolate its results to large storm tank real cases which are comprised of a large number of such individual elements.

The study of such elements takes place in three different stages. In the first one, it is characterizing behaviour of them by CFD (Computational Fluid Dynamics) techniques. Such characterization basically involves the determination of each energy dissipation element according to the flow when water flows in either of directions in which these elements can be installed. Characterization is completed with the study of influence on the energy dissipation of these modular elements that suppose the arrangement of adjacent modules in both directions. To this end, this study is developed through CFD techniques and considering modular elements are disposed in both, longitudinal and transverse flow directions.

In the second stage, the validation of analyses performed by CFD techniques takes place. For this purpose, computational results are contrasted with results obtained from a canal prototype built in hydraulic laboratory of Los Andes University in Colombia.

In last stage, from both computational and experimental studies a one-dimensional behaviour model of these modular elements is developed with the main purpose of integrating it into specific computational models of drainage networks analysis as SWMM [2].

3. Methodology and case study.

To characterize hydraulic behaviour of these modular elements, CFD techniques are to be used. These ones have already been used in the field of water management [3]; specifically, in urban drainage systems [4] and to characterize deposits as well [5]. Particularly, to solve this issue the model ANSYS® Fluent [6] is used in a simplified geometry of the element that has been created from completely defined one thereof by CAD solids representation model.

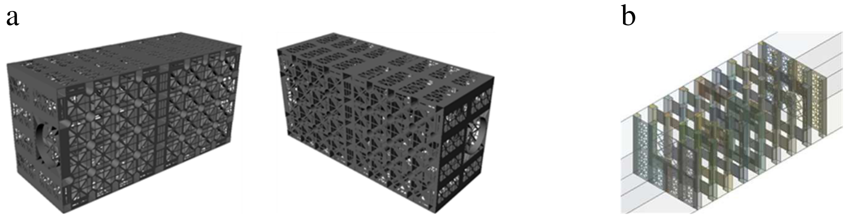

Fig. 2. (a) Actual geometry and (b) simplified geometry of the modular structure.

The different key stages followed in analysis by using CFD techniques to define the behaviour of these structures are: geometry definition, meshing, processing and post-processing or results presentation.

These four stages are not confined to a purely sequential process. It is rather an iterative process in which from the results (post-processing) should be considered whether it is necessary to modify simulation parameters (processing), the size of elements the workspace is divided (meshing) or even change the shape model (geometry).

From fully defined geometry, a simplified model that represents accurately existing relevance geometric characteristics along modular structure in the same flow direction is performed. In addition, it is important to emphasize that simplified model has all sheet members without thickness in order to ease its meshing, reducing number of its cells and therefore increasing its calculation speed.

The meshing model is selected trying to respect fully the regular hexahedral shape of cells to ensure a good mesh quality, with asymmetry values very close to 1.

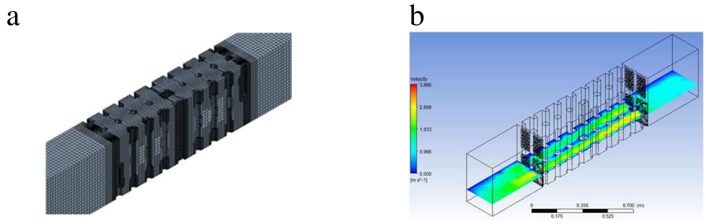

Fig. 3. (a) Meshing and (b) post-processing CFD analysis of a modular block.

As for the mesh, analysis of water behaviour inside modular blocks is performed using VOF (Volume of Fluid) model, which was developed by [7] to determine the amount of both water and air phases that present each of cells in the model. In order to completely define the flow a turbulence k-ε model [8] is used for being the most suitable one for the turbulence levels and Reynolds number values that are reached inside this sort of structures.

The resolution numerical scheme used in resolving the problem is the “pressure based” type, using a “simple” type scheme for the coupling between pressure equations and velocity ones.

Based upon this model configuration and input flow definition, the analysis is performed. Simulation runs for “transient regime” and it concludes when “permanent regime” is established in the system. Once calculation process is completed one proceeds to results post-processing Fig. 3b, that allows to analyse the flow characteristics developed within modular structures as well as to determine depth profiles along the models studied. From which will be possible to obtain Manning coefficient (n) and head minor losses dimensionless (k) values, these parameters characterize energy dissipation produced by the mentioned modular elements existence.

4. Configurations analysed.

Once CFD model has been built is necessary to define the case studies will proceed to address. Since the aim pursued is to determine energy dissipation of these modular blocks in both directions. Firstly, the intention is representing the behaviour of an isolated single modular element to determine the influence that presents the installation of several of these elements presents one after the other in both directions afterwards.

Table. 1: Detail of different configurations to study.

Configuration	1x2x1	1x2x4	2x2x1	2x2x4	1x2x1T	1x2x8T
No rows of modular blocks in flow direction	1	4	1	4	1	8
No rows of modular blocks in perpendicular direction to the flow.	1	1	2	2	1	1
No rows of modular blocks in vertical	2	2	2	2	2	2
Total number of modular units installed	2	8	4	16	2	16

Finally, two different cases are analysed. The first one is based on studying the behaviour of a single modular element individually located in flow direction. It allows to carry out a detailed analysis of located head losses at both, entrance and exit of the modular block as well as losses distributed in a uniform way along the modular element itself. The second case includes a series of four consecutive blocks. Both cases are analysed through CFD techniques and results obtained by this modelling methods have been validated on the basis of flow profiles obtained from an experimental prototype built at laboratory placed at Los Andes University in Colombia.

Profile obtained from CFD simulation for the first case described in previous paragraph for a circulating flow of 37.83 l/s is shown in Fig. 4.

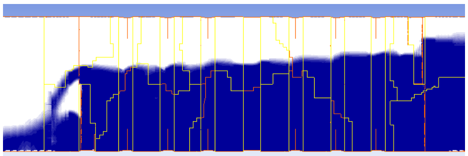

Fig. 4. Detail of profile adopted by water flow along the modular block in 1x2x1 configuration, Q=37.83 l/s.

Then, it proceeds to validate profile obtained from CFD simulation with results observed in laboratory. Comparative profiles of both can be seen in Fig. 5. Observing Fig. 5 it is easy to conclude that a significant similarity exists between two profiles drawn. Therefore, it is assumed as valid way of proceeding conducted for the solution by computer modelling. Subsequently, calculation of a computer simulation series for different flow rates are performed for the same configuration, using an identical procedure. Profiles obtained for all flow rates tested are presented in Fig. 6.

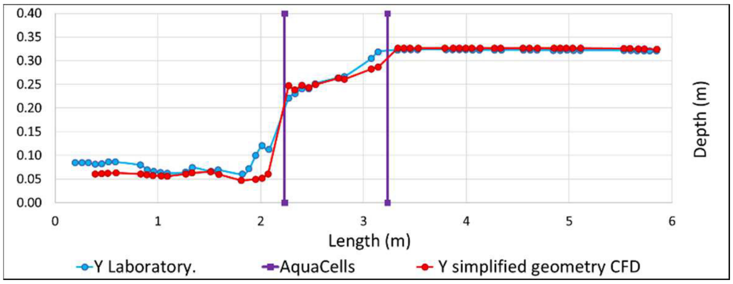

Fig. 5. Comparative profiles obtained in laboratory vs CFD in 1x2x1 configuration, Q=37.83 l/s.

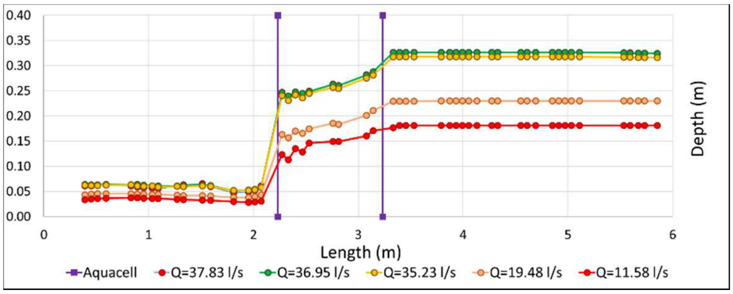

Fig. 6. CFD comparative profiles obtained for different flow rates in 1x2x1 configuration.

Once CFD results are validated proceed to determine the parameters that characterize modular block behaviour assimilating it to a pipe. In this case, it is intended to determine Manning coefficient values and pressure losses at the entrance and exit of the modular structure. To do this is taken as starting point depth values at different points located just before and after sides of Aquacell. The value of dimensionless head minor losses coefficient on one side of the Aquacell is determined by analysing the energy loss between a point before (point 1) and one after (point 2) of said face. Mathematically “k” coefficient is obtained though the expression (1).

$$k=\frac{2g\cdot\Delta H}{\left(\frac{v_1+v_2}{2}\right)^2} \tag{1}$$

In expression (1):

- k is the value of the located head losses dimensionless coefficient in a particular face of the Aquacell.
- v_1 y v_2 are velocity values at upstream and downstream points of both, input and output modular block faces.
- ΔH is the energy loss between points 1 & 2. This energy gap is obtained according to the expression (2).

$$\Delta H=E_1-E_2=\left(z_1+\frac{v_1^2}{2g}\right)-\left(z_2+\frac{v_2^2}{2g}\right) \tag{2}$$

Similarly, equivalent Manning coefficient is calculated for the stretch of the Aquacell, and it is obtained considering an average value from input and output velocities as well as an average one from hydraulic radius. The slope of the hydraulic grade line “s” is obtained as a ratio among the energy gap between inlet and outlet of Aquacell and the length “L” between these points. This calculation approach is applied in two cases: to the results obtained directly from laboratory tests and to which have been achieved by CFD techniques applied to simplified geometry included in the model (Table. 2).

Table. 2: Located head losses and equivalent Manning coefficient of the Aquacell from CFD results in 1x2x1 configuration, Q=37.83 l/s.

Q = 37.83 l/s – CFD results for simplified geometry.													
Obtaining dimensionless k loss coefficient													
Q	z1	z2	v1	v2	$v1^2/2g$	$v2^2/2g$	E1	E2		ΔH		ki	
37.83	0.33	0.29	0.23	0.26	0.00	0.00	0.33	0.29		0.04		12	
37.83	0.25	0.06	0.31	1.24	0.00	0.08	0.25	0.14		0.11		4	
Obtaining equivalent Manning coefficient.													
Q	x1	z1	x2	z2	L	Dz	s	v1	v2	A1	A2	Rh	n
37.83	-3.142	0.286	-2.48	0.24	0.68	0.044	0.065	0.26	0.312	0.14	0.12	0.13	0.225

For different flow rates studied, in Table. 3 are shown values of equivalent Manning coefficient (n_{eq}) as well as located loss ones at inlet (k_{in}) and outlet (k_{out}) of the modular structure arranged in the same direction of the flow in 1x2x1 configuration.

Table. 3: n-k values obtained by CFD techniques for different flow rates in 1x2x1 configuration.

Q (l/s)	CFD results for simplified geometry		
	n_{eq}	k_{in}	k_{out}
37.83	0.225	12	4
36.95	0.229	13	4
35.23	0.238	13	4
19.48	0.252	16	5
11.58	0.220	22	6

In case of four consecutive blocks existence, the profile obtained for a 37.83 l/s circulating flow rate is presented in Fig. 7.

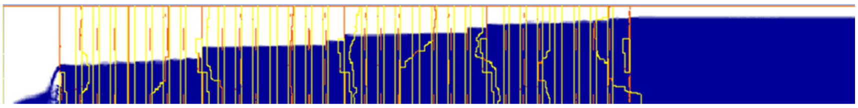

Fig. 7. Profile adopted by water flow in 1x2x4 configuration, Q=37.83 l/s.

In Fig. 8 the validation graph for the computational solution is shown. In which, profile obtained by CFD is compared with laboratory measured one. As a final result, CFD's solution is considered valid for 1x2x4 configuration. In the same way as discussed for 1x2x1 configuration, it proceeds to calculate Manning coefficient values and located losses "k" dimensionless ones for each of modular block for both cases laboratory and computationally modelled one. Values obtained appear in Table. 4.

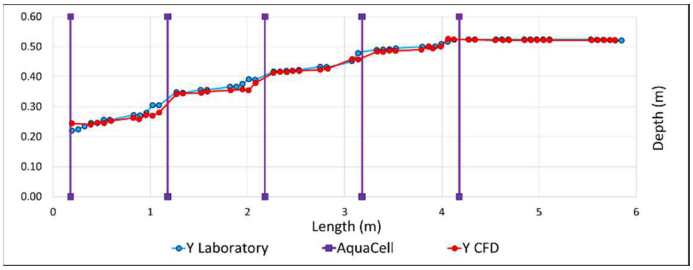

Fig. 8. Comparing profiles obtained in laboratory vs CFD in 1x2x4 configuration, Q=37.83 l/s.

Table. 4: n-k values comparison between laboratory and CFD techniques in 1x2x4 configuration, Q=37.83 l/s.

Modular blocks order	Laboratory		CFD	
	n_{eq}	k_i	n_{eq}	k_i
1	0.279	21	0.318	20
2	0.372	28	0.252	40
3	0.256	27	0.193	27
4	0.226	23	0.207	20

In this case, “k” coefficient of block 1 only includes losses at its entrance while for the other ones, value of “k” is the sum of located losses produced on the outlet of previous block and losses generated at the entrance of preceding one.

5. One-dimensional model of a modular storage.

As discussed above in the section on objectives, the aim of this study is to make a proposal of behaviour’s representation of this kind of modular structures by one-dimensional models that can be integrated into specific computational models of drainage networks analysis as is the case of SWMM.

From the model developed in SWMM has been made the adjustment of the values of both, the Manning coefficient (n) and head minor loss dimensionless coefficient in the inlet (k_{in}). This adjustment has been made based on a computational tool using SWMM and testing for each flow rate is what are the coefficient values that best fit.

After making the adjustment for the different flows which have been simulated by means of CFD and in order to show the goodness of the results achieved by adjusting n_{eq} and k_{in} coefficients, a comparative graph between results from SWMM simulation and computationally calculated ones can be observed in Fig. 9. In this graphic image can be appreciated points corresponding to energy flow values along the four consecutive modular elements disposed in the same direction of the flow in this configuration and obtained by CFD techniques, on which a green line connecting points between consecutive elements and corresponding to energy values obtained for the best combination of parameters found is overlaid.

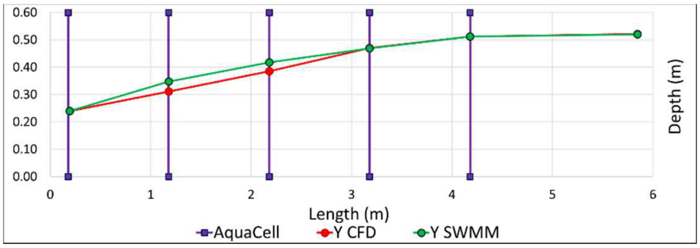

Fig. 9. Profile’s comparison between CFD vs. optimal parameters ones for 1x2x4 configuration, Q=37.83 l/s.

Table. 5: Comparison of optimal n_{eq} and k_{in} parameters obtained from laboratory vs CFD in 1x2x4 configuration.

	Laboratory		CFD	
Q(l/s)	n_{eq}	k_{in}	n_{eq}	k_{in}
37.83	0.05	40	0.01	37
30.37	0.01	45	0.04	34
25.59	0.04	41	0.04	34
21.61	0.27	25	0.04	35

Finally, in Table. 5 are presented values of n-k parameters obtained from SWMM both for the case tested in laboratory and simulated in Ansys for all cases studied.

6. Conclusions.

The main conclusion of this study is that CFD models developed are absolutely valid to analyse and study both the behaviour as well as energy dissipation of these modular structures used to build detention tanks. All models have shown a good correlation with experimental model data. This models have allowed to established perfectly the energy dissipation of such elements. Despite its complexity, this dissipation can be considered it is concentrated in three different points thereof; in the module inlet, in its outlet and a distributed dissipation along all module length. Moreover all analysis and studies done for both experimental and CFD models have allowed to build a one-dimensional characterization model of such structures, which has been implemented in the hydraulic SWMM model.

In sum, CFD techniques have proven to be highly effective in the study and characterization of modular elements. Results obtained from computer simulations and experimental verification ones are fairly close. Thereof, it can be said that these techniques can be used firstly as a tool to characterize larger detention tanks. In addition, such a tool can be used to determine the energy dissipation of these structures in different flow conditions as well as in other configurations to those analysed without the need to spend the enormous amount of money that laboratory experiments imply. Certainly, its use in more depth in the future will allow to analyse larger detention tanks built with this kind of modular structures and the determination of what conditions are like filling and emptying them when heavy rain episodes will occur.

Acknowledgements.

Authors acknowledge Mexichem Colombia for funding this research through the Project “Urban drainage and climate change: to sewerage systems of the future. Phase II”, without which would have been impossible to complete these studies.

References.

[1] Andrés-Doménech, I. ; Montanari, A. ; and Marco, J.B. (2012). "Efficiency of Storm Detention Tanks for Urban Drainage Systems under Climate Variability". *Water Resour. Plann. Manage.* 2012.138:36-46.

[2] Rossman, L.A. (2015). *Storm Water Management Model. User’s Manual Version 5.1.* EPA, Cincinnati, OH (EEUU).

[3] Jarman, D. S., Faram, M. G., Butler, D., Tabor, G., Stovin, V. R., Burt, D., & Throp, E. (2008). “Computational fluid dynamics as a tool for urban drainage system analysis: A review of applications and best practice”. *11th ICUD – 11th International Conference on Urban Drainage.* 31 August - 5 September 2008, Edinburgh, Scotland.

[4] Jarman DS, Faram MG, Tabor G, Butler D. (2007). “A review of the opportunities presented through the application of Computational Fluid Dynamics (CFD)to water management challenges”. *Water Management Challenges in Global Change. CCWI2007 and SUWM2007.* Leicester, UK.

[5] Adamsson, Å., Bergdahl, L., & Lyngfelt, S. (2005). “Measurement and three-dimensional simulation of flow in a rectangular detention tank”. Urban Water Journal. 2(4), 277-287.

[6] ANSYS Inc. (2013). *ANSYS Fluent User's Guide. Release 15*.0. ANSYS Inc, Southpointe, Canonsburg, PA (EEUU).

[7] Hirt, C.W.; Nichols, B.D. (1981). "Volume of Fluid (VOF) method for the dynamics of free boundaries". *Journal of Computational Physics* 39 (1): 201–225. doi:10.1016/0021-9991(81)90145-5.

[8] Launder, B. E., & Shima, N. (1989). "Second-moment closure for the near-wall sublayer. Development and application". *AIAA journal*, 27(10), 1319-1325.