Improvement of a system for catchment, pre-treatment and treatment of runoff water using PIV tests and numerical simulation

M. Alonso-Martínez[1], A. Navarro-Manso[2], D. Castro-Fresno[3], F.P. Álvarez-Rabanal[4], J. J. del Coz Díaz[5] *

[1] *GICONSIME Research Group, Department of Construction and Manufacturing, University of Oviedo, 33203 Gijón (Spain)*

[2] *Department of Energy, University of Oviedo, 33203 Gijón (Spain).*

[3] *GITECO Research Group, Dept. of Transport and Project Management, University of Cantabria, 39005 Santander (Spain)*

[4] *Department of Construction and Manufacturing, University of Oviedo, 33203 Gijón (Spain)*

[5] *Department of Construction and Manufacturing, University of Oviedo, 33203 Gijón (Spain)*

ABSTRACT

This paper studies how to improve the efficiency of a new System for Catchment, Pre-treatment and Treatment of runoff water (SCPT). This system is integrated into an urban sustainable gravity settler which can decrease diffusive pollution. This study provides important advantages for the ecosystem by improving new sustainable drainage to clean runoff water. In this research work, an investigation methodology known as Hybrid Engineering (HE) was used. HE combines experimental tests and numerical simulations, both of them conducted on a 1:4 scale prototype. In this study, numerical simulations by the Finite Volume Method (FVM) and experimental tests by Particle Image Velocimetry (PIV) were compared. A strong correlation between the numerical and experimental analysis was found. Next, the efficiency

---

* Corresponding author. Tel.: +34-985-182042; fax: +34-985-182433.
E-mail address: juanjo@constru.uniovi.es (J. J. del Coz Díaz).

of the SCPT was optimized by Design of Experiments (DOE). Analysis of experimental and numerical results and their comparison are presented in this research paper.

Keywords: Computational Fluid Dynamics (CFD); Design of Experiments (DOE); Particle Image Velocimetry (PIV); Sedimentation; Volume modelling; Vortex dynamics.

INTRODUCTION

The pollution of runoff water is an important problem which seriously affects the environment. The poor quality of storm water which is filtered by the terrain causes contamination known as "diffusive pollution". Several solutions for this problem have been studied since the nineteenth century (Caltrans 2010; Campbell *et al.*2004; Castro Fresno *et al.* 2005; Dolz and Gómez 1994; Novotny 2003). In these previous studies, BMPs (Best Management Practices) were developed and the efficiency of the new devices was discussed. As a conclusion of these projects, it was proved that the systems used are not enough to improve the water quality. In this sense, Fernández Barrera in his Doctoral Thesis in 2009 and other authors developed a new System of Catchment, Pre-treatment and Treatment (SCPT) of the pollutants of the runoff water. This new system was patented (Castro Fresno *et al.* 2010).

The SCPT consists of two sections divided by a flat panel, which is also called the screen. The function of the first of these sections is as a hydraulic plug where it is possible to retain oils. Then the water flows under the flat panel and goes inside the second section which works as a gravity settler. In this second section, geotextil layers can be included in order to filter the water. This system allows the reduction of the environmental impact due to the contaminants of the runoff water, one of the most important problems of the storm water.

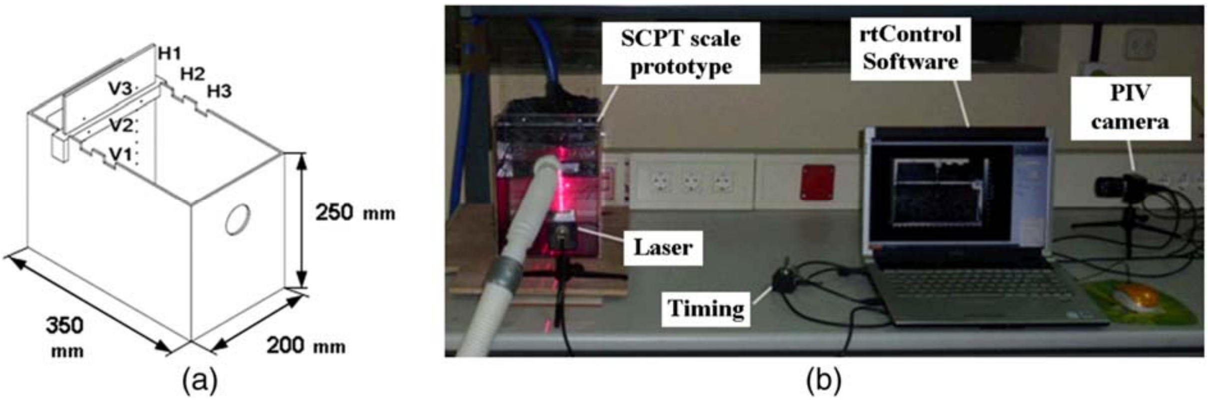
This system was tested in a long-term laboratory simulation in order to study its performance. The SCPT is able to avoid resuspension of contaminants which is an important problem in other devices. The conclusions of this paper consider the efficiency of the SCPT taking into account two kinds of pollutants, solids and oils. A review of other Sustainable Urban Drainage Systems (SUDS) has been published where the main sustainable drainage practices in Spain are commented (Castro-Fresno *et al*. 2013).

GITECO research group has continued the investigation of the SCPT carrying out a laboratory analysis of a 1:1 scale prototype, Fernández Barrera 2009; Rodríguez-Hernández *et al*. 2010; Fernández-Barrera *et al*. 2010). Three parameters were studied in that research work: inflow, pollutant loads, considering solids and oil in water and the set up of the filtration system. This study has shown the influence of these parameters on the SCPT efficiency. The results have given high efficiencies of runoff water treatment: 85% for solids pollutants and 97% for oil in water.

FVM was already used in previous research works about the SCPT (del Coz Díaz *et al*. 2011). Numerical simulations were carried out to study the performance of the SCPT using a simplified geometrical model. Furthermore, an optimization of the SCPT taking into account the position of the flat panel and the particles diameter as the input parameters and the SCPT efficiency as the output parameter was undertaken. The consistent results obtained in this investigation indicate that FVM is an adequate tool to study this system. Another important conclusion obtained in this numerical analysis was the large influence of the panel separation on the efficiency of the SCPT. Finally, in this previous study the final results are kept open to experimental validations in order to compare both results, numerical and experimental.

Summarizing, the SCPT is an important contribution to environmental engineering because it is able to treat storm water and clean it before it filters into the terrain. For this reason, an in depth study of the system's behaviour has been developed in this paper in order to increase its efficiency. The methodology used to complete the SCPT study is based on a combination of the experimental and numerical studies. This technique is known as Hibrid Engineering (HE) and has been used in many other fields too (Flamant *et al*. 2004; Mayurkumar *et al*. 2012).

In this work, the influence of the flat panel of the SCPT on its efficiency has been studied. A 1:4 scale structure of the SCPT, made of methacrylate, has been studied. This structure is 0.2m wide, 0.35m. long and 0.25m. high. The flat panel has been studied in several positions, moving it in vertical and horizontal directions. The scale prototype and the different positions studied are showed in Figure 1. Experimental scale tests have been developed with this prototype using Particle Image Velocimetry (PIV). Furthermore, numerical simulations have been carried out using the Finite Volume Method (FVM) and the best flat panel configuration has been selected by means of an optimization using the Design of Experiments technique (DOE). Finally, experimental scale tests have been developed using the Particle Image Velocimetry (PIV) technique.



(a) (b)

Figure 1. SCPT scale prototype studied: (a) dimensions and panel positions; (b) general assembly of the laboratory tests.

The aim of this research paper is to complete the previous investigations about the SCPT, considering three advanced techniques: numerical simulation by FVM, optimization using DOE and experimental measures by PIV.

Techniques used

Two different techniques have been used in this study. On the one hand, the PIV technique has been used to measure the speed of the fluid inside the SCPT scale model. Several parameters can be obtained using this technique which has been studied by different authors in several fields (Melling 1997; Raffel *et al.* 1998; Schroeder *et al.* 2008; Wang *et al.* 2009). On the other hand, a numerical method based on FVM has been used to analyze the behaviour of the system. Furthermore, DOE has been used to optimize the position of the screen and improve the efficiency of the SCPT. The combination of FVM and DOE has been used for years (Zienkiewicz *et al.* 2005; Madenci and Guven 2007; del Coz *et al.* 2007).

EXPERIMENTAL TESTS

The PIV technique was used in order to study the performance of a small scale SCPT. This method has allowed the measurement of the fluid velocity inside the SCPT.

The equipment used in the laboratory tests was a laser PIV system "H41" (Etalon Research 2009). The PIV technique has been used for measuring the fluid speed at multiple points throughout a 2D plane. The fluid is seeded with particles which follow the fluid movement inside a volume. It is possible to deduce the movement of the underlying flow by tracking the motion of these particles. This technique is well explained and studied by other authors (Wereley and Gui 2001; Van Hooff *et al.* 2012).

A laser illuminates the SCPT, and the camera, which is normal to the illuminated plane, captures the particle velocity in that section. An electronic control allows pulse separation control between 100 μs and 5 s, and width pulse control between 10 μs and 32 ms. The software processes the velocity map by cross-correlation using capture imaged in each laser pulse.. Furthermore, the equipment has an optical standard system which is able to make a light plane which is 3x200x250 mm (width x height x length). This optical system can be used with two angular options, 22º or 45º; in this study only 45º opening has been used. The resolution of the camera is 640 x 480 with digital output format of 8-bit and the pixel size is 6 x 6 μm ($8.5 \cdot 10^{-2}$ m format).

The frequency of this camera is 16 Hz, with a standard photographic lens of $1.25 \cdot 10^{-2}$ m, 1:1.4, and is able to register distances between 0.3 and 2 m.

The images captured are processed by "rtControl Software", which was provided by the equipment manufacturer. This software takes the captured data, divides the images in sub-windows and then analyzes consecutive images by cross-correlation. The time between sub-windows is known and so, the velocity map can be obtained. It is important to take into account the influence of external factors which can reduce the quality of the images, such as the kind and size of the seed particles, the reflective surfaces, etc. In this sense, due to the complexity of the PIV technique the experience of the authors is very important to obtain success results.

This software can also provide other parameters like the velocity vector (in real time) of the particles in a fluid, and statistical data like vorticity, average speed, etc.

Set up of the laboratory tests

The configuration of the PIV tests has taken into consideration the following aspects:

- The seed particles, as suggests the PIV equipment manufacturer, are polyamide particles of 100 µm, whose density is the same as water density. The polyamide particles have suitable optical properties in order to be illuminated by the laser.
- The superposition of images is not possible. So, the captured plane by the camera must be smaller than the illuminated plane by the laser. In this case, the maximum plane size that the camera can capture is 0.5 m. long. In this case, a 1:4 scale SCPT has been used, whose 2D plane captured is 0.35 m long.
- The scale model inlet is a longitudinal groove in a cylindrical pipe whose volume of water is about 0.1 l/s in order to obtain a non turbulent inflow. The inlet is immersed to avoid the turbulence and obtain a regular flow inside the system. Besides, the seed particles and the water have been mixed before the inlet of the SCPT to make it easier to follow the water movement.
- The equipment calibration must be done before starting the measures by PIV. This calibration assigns units to the results of the software. It is important to know that only the results which have been taken during the same calibration can be compared.
- Several positions of the flat panel have been studied to obtain experimentally the influence of the screen displacement over the SCPT efficiency, see Figure 1.

The laboratory test assembly is shown in Figure 1,where the laser illuminating the SCPT model is shown on the left side. Also, the camera normal to the illuminated plane and the software used are shown on the right side.

With respect to the PIV equipment configuration, the parameters of the camera and the laser must be carefully chosen to obtain accurate captures of the fluid movement. These parameters must be adapted to the fluid velocities inside the scale SCPT.

The maximum velocity of the fluid in the SCPT is at the inlet. This value has been calculated based on the maximum inlet volume and the minimum inlet surface. The maximum velocity obtained was below 0.2 m/s. The calibration in this work has been 3.5 mm/pixel. The fastest particles, therefore, cannot travel more than ¼ of the sub-window size between images. So, the values of the pulse separation, Δt, and the pulse width, δt, have been estimated accordingly with equation (1).

$$\Delta \mathrm{t} = 1/4 \cdot 5 \cdot \mathrm{S_w} / \mathrm{V_{max}} \cdot 10^3 \qquad (1)$$

Δt: pulse separation (s)

S_w: Sub-window size (pixels)

V_{max}: Maximum velocity (m/s)

Values of pulse width depend on values of pulse separation according to equation (2).

$$\delta \mathrm{t} \;\Box\; \Delta \mathrm{t} \cdot 10^{-3} / 10 \qquad (2)$$

δt: pulse width (s)

In addition to the pulse separation and pulse width there are other parameters of configurations in the PIV camera that depend on the boundary conditions of each test. The values of these additional parameters are provided in Table 1.

Table 1 Values of the configuration parameters for the PIV camera.

Parameter	**Value**
Δt	58.22 (ms)
δt	4.27 (ms)
Step	Single
Sub-window size	12 px
Overlap	50%
Vector lenght	10 (mm)
RMS	0.6

Six different tests have been carried out, one for each flat panel position inside the scale model of the SCPT. For horizontal displacements the distances between the inlet wall and the panel have been 120, 145 and 170 mm, wich are H1, H2 and H3 respectively. For vertical movements the distances between the end of the panel and the bottom of the SCPT have been 70, 90 and 110 mm, wich are V1, V2 and V3 respectively.The first horizontal position, called H1, cannot be tested due to its proximity to the inlet fluid.

Taking into account previous research works focused on the flow simulation and sediment transport over channels (Penko and Calantoni 2013), as well as previous full-scale experiments over the SCPT (Fernandez Barrera 2009, Fernandez Barrera *et al.* 2010, del Coz *et al.* 2011) the vorticity was considered an important result. The rotational fluid movement inside the device and the conclussions derived from Penko and Calantoni provide the importance of the vorticity values in this specific case.

In conclusion, the main result obtained in the experimental test was the average vorticity value, W. This result was calculated as the scalar component of the angular velocity vector or punctual vorticity, ω, for a two-dimensional input, in this case H and V, which corresponds to the plane illuminated by the laser. The following equation (3) was used (Nezu and Nakagawa 1993):

$$\omega = \frac{dv}{dx} - \frac{du}{dy} \quad ; \; W = \frac{\sum_{1}^{m\times n} \omega_i}{m \times n} \qquad (3)$$

Where:

u and *v* are the velocity components with respect to *x* and *y* directions.

m and *n* are the grid in x and y directions, respectively.

The scalar punctual vorticity, ω, was calculated at each grid point, $m \times n$, of a fine mesh of 3.5 mm size that fill the capture of the illuminated plane. In order to avoid local outlet suction, which may change the flow path, the experimental data were selected. In this way, we have removed 50 mm at the end of the output chamber, corresponding to 12 to 14 horizontal grid points approximately.

NUMERICAL SIMULATIONS

The numerical simulation presented in this paper has been carried out by the CFX module in the ANSYS Workbench software v.12.1. This work has been supported by the authors’ experience in the field of simulation and non-linear analysis. Over several years, the authors have worked using finite element methods (FEM) and finite volume method (FVM), (del Coz *et al.* 2007; del Coz *et al.* 2011; García Nieto *et al.* 2010). These methods provide great results when the models are correctly compared and validated.

The relationship between the horizontal position of the flat panel and the particle collection efficiency has been predicted in del Coz Diaz et al. 2011. However, in this paper a new numerical model has been developed in order to study the behaviour of the SCPT from the velocity field point of view. In this sense, the input parameters previously used have been studied in both experimental and numerical ways. In summary, the contribution of this paper is important in order to prove the ability of the PIV and numerical methods to verify the SCPT performance.

Finite volume model

In this sub-section the finite volume model used in this investigation is explained:

- Geometrical model: the geometrical model which has been used in the SCPT numerical simulations reproduces the fluid volume inside the system. The dimensions of the geometrical model used in the numerical simulation and the experimental SCPT are the same.
- Finite volume types: the meshing of the geometrical model by finite volumes has been carried out using the "Automatic Method" in ANSYS. This method uses a patch conforming algorithm for tetrahedrons method control by a Delaunay tetra mesher. If possible, the mesher builds tetrahedrons volumes and tries to create a smooth size variation based on the specified growth factor. In this case, the volume size has been established between $2\cdot10^{-3}$ and $6\cdot10^{-3}$ m. The result has been a mixed mesh with mainly tetrahedral and hexahedral volumes, although others types of finite volumes have been included in the interface areas. The geometrical model consists of 389,265 elements and 110,066 nodes. The majority of the element quality values are about 0.75, what means a good mesh quality (Zienkiewicz *et al*. 2005; Madenci and Guven 2007).
- Boundary conditions and loads: the fluid flows by the input chamber, then it goes through the output chamber and finally it goes out the system by the outlet. The results of the numerical simulations have been considerably influenced by the configuration of

the transition area. The interface areas have been located between the inlet and input chamber, between the input chamber and output chamber and between the output chamber and outlet.

This finite volume model has been solved by an iterative process to obtain the solution of the problem.

Numerical simulation based on DOE

The influence of the screen displacement on the SCPT performance has been studied in order to improve the efficiency of this system. In this sense, an optimization procedure based on the Design of Experiments technique (DOE) on the FVM models has been carried out in order to know the best position of this screen from the efficiency point of view (del Coz *et al*. 2013; Montgomery 2001).

The input parameters of the DOE technique have been the horizontal and vertical positions of the flat panel. The output parameter in the DOE has been a derived parameter, which indicates the performance of the system. The efficiency of the system has been obtained from the reduction of the average vorticity inside the SCPT. If the vorticity of fluid flow in the input chamber is high, the majority of the solid particles will stay inside the input chamber (Penko and Calantoni 2013). The few particles that could go to the output chamber will precipitate at the bottom of the SCPT due to this chamber working as a gravity settler. A large difference in the average vorticity between the input and the output chamber means that the SCPT performance is good.

In this case, the punctual vorticity, ω, was calculated by means of the equation (3) at each finite volume taking into account the meshing parameter between $2\cdot10^{-3}$ and $6\cdot10^{-3}$ m. Next, the average vorticity, W, was obtained as the mean value of the punctual vorticity for all finite volumes in each camera. Furthermore, the dimensions of the input and output chambers are changed according to the position of the flat planel as has been studied in the DOE analysis.

The parameters used in the analysis by DOE are the following:

- Input parameters:
    - Horizontal position: from 70 to 170 mm, with an initial value of 120 mm.
    - Vertical position: from 70 to 120 mm, with an initial value of 95 mm
- Output parameters:
    - Average vorticity in the input chamber
    - Average vorticity in the output chamber
    - Vorticity reduction in the SCPT

The technique which has been used in DOE is Central Composite Design (CCD). The limited values used in the DOE technique are determined by the SCPT dimensions. The maximum and the minimum positions of the flat panel are taken as boundary values. The initial conditions are intermediate values. The total CPU time for each numerical model has been $7.955 \cdot 10^3$ seconds (2 hours, 12 minutes and 35.281 seconds).

The reduction of the vorticity is defined by equation (4).

$$R = \left(W_{ic} - W_{oc}\right) / W_{oc} \cdot 100 \qquad (4)$$

While:

R: Vorticity reduction inside the SCPT

W_{ic}: Average vorticity in the input chamber [rad/s]

W_{oc} : Average vorticity in the output chamber [rad/s]

The vorticity reduction represents the efficiency of the system and indicates a decrease of the fluid rate inside it. So, the best efficiency is obtained with maximum vorticity reduction. The best position of the flat panel to obtain a high level of efficiency is shown in the response surfaces obtained by DOE.

EXPERIMENTAL AND NUMERICAL RESULTS

Laboratory test results

The results of the laboratory tests provide information about the fluid trajectory inside the SCPT and the velocity map of the fluid.

The PIV equipment gives results in several formats. In this case, MATLAB® files and .avi files were evaluated. The.avi files provide qualitative results and the Matlab files provide quantitative results.

In Figure 2, the results from the laboratory tests of the scale prototype are shown. The fluid inlet is on the right and the outlet on the left. Different colours are used in these representations to show the different velocities, from blue for lower velocity to red for higher velocity.

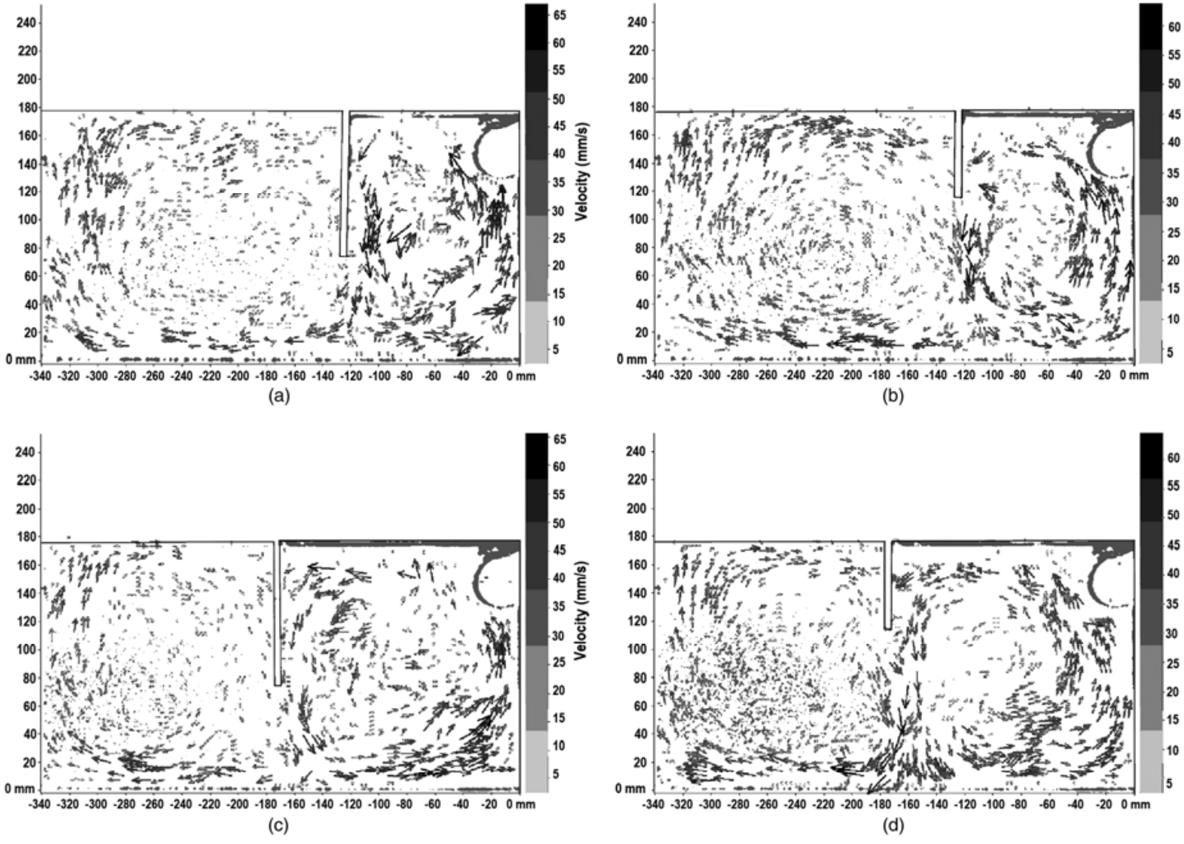


Figure 2. Screen captures in the PIV tests for different flat panel positions: a) H2 – V2; b) H2 – V3; c) H3 – V2; d) H3 – V3.

The behaviour of the fluid in the SCPT prototype is depicted in Figure 2. The fluid velocity inside the input chamber is high. The majority of the solid particles in this volume are rotating with the fluid. The output chamber is longer than the input chamber in order to decrease the fluid rate. The fluid goes into the output chamber containg very few solid particles, which precipitate at the bottom of the SCPT due to its low velocity. Finally the fluid which goes out is free of solid contaminant particles, and the particles at the bottom of the system are removed by a drain.

As it is shown in Figure 2, the fluid has two vortices, one in the input chamber and other in the output chamber.

To obtain quantitative results, the Matlab files were analyzed. A velocity matrix is taken every 0.1 s. After several seconds, all matrixes are combined by software in order to represent the velocity map of the fluid from experiments. These results are analyzed by a matlab code and incoherent vectors are deleted. Then, the punctual vorticity parameter, ω, is obtained in each matrix, along with the average vorticity, *W*, for each test. The velocity map obtained in the experiments is plotted in Figure 2.

The vorticity or rotor vector of velocity is the same concept as the angular velocity of a fluid. The vorticity was considered as the best parameter to represent the rotational behaviour of the fluid in the SCPT. The average vorticity value in the input chamber and in the output chamber is different (see Table 2). Improved performance in the SCPT is given by a considerable vorticity decrease between both chambers. If the average vorticity is high, as in the input chamber, the solid particles are rotating inside the fluid. The few particles that go to the output chamber precipitate because the average vorticity in this volume is low. The fluid that goes out of the system is clean and without solid particles, and in this way, the runoff water that is filtered to the terrain is less contaminated.

Table 2. Average vorticity values obtained in the laboratory tests.

Screen position (mm)		**Vorticity (rad/s)**	
Vertical	**Horizontal**	**Input chamber**	**Ouput chamber**
70	120	0.5337	0.1154
	145	0.5051	0.1519
	170	0.4582	0.1724
90	120	0.5268	0.1298

	145	0.4951	0.1765
	170	0.4572	0.2267
110	120	0.5200	0.1531
	145	0.4848	0.2000
	170	0.4386	0.2880

In order to compare the laboratory test and the simulation results the average vorticity evolution is obtained. Desired performance of the SCPT indicates a negative trend which means that the vorticity between the input and the output chamber is decreasing.

Numerical results

On the one hand, the results obtained in the numerical simulations by FVM are shown in Figure 3.

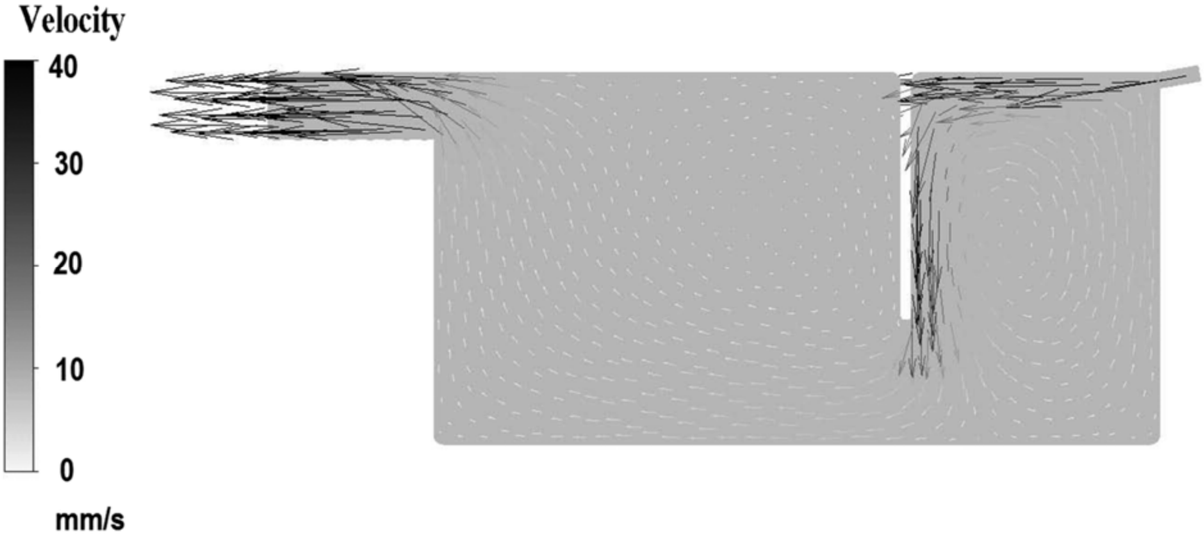


Figure 3. Numerical simulation results: velocity vectors map in the SCPT for a flat panel position of H = 120 mm. and V = 90 mm.

The numerical simulation shows that the fluid inlet is at a high velocity, on the right hand side in Figure 3. The velocity in the output chamber decreases as it is shown in Figure 3 on the left hand side. The average vorticity in the SCPT and its trend are the results of the numerical simulations.

Due to the "section assignment" previously described in the numerical process, two independent analysis have been conducted in both chambers. The graphic results show the fluid rate decreasing between the input chamber and the output chamber. For the flat panel position studied in this numerical simulation, the maximum vorticity values obtained are $42.95 \cdot 10^{-2}$ rad/s and $14.39 \cdot 10^{-2}$ rad/s, in opposite directions as depicted in Figure 3. The efficiency of the SCPT for these results is about 66.5%, which is obtained from the reduction of the fluid rate.

On the other hand, the influence of the input and output parameters can be seen in the response surfaces obtained by the DOE technique. Mainly, three different output parameters have been studied: the input and the output chamber average vorticity, as well as the vorticity reduction between both chambers.

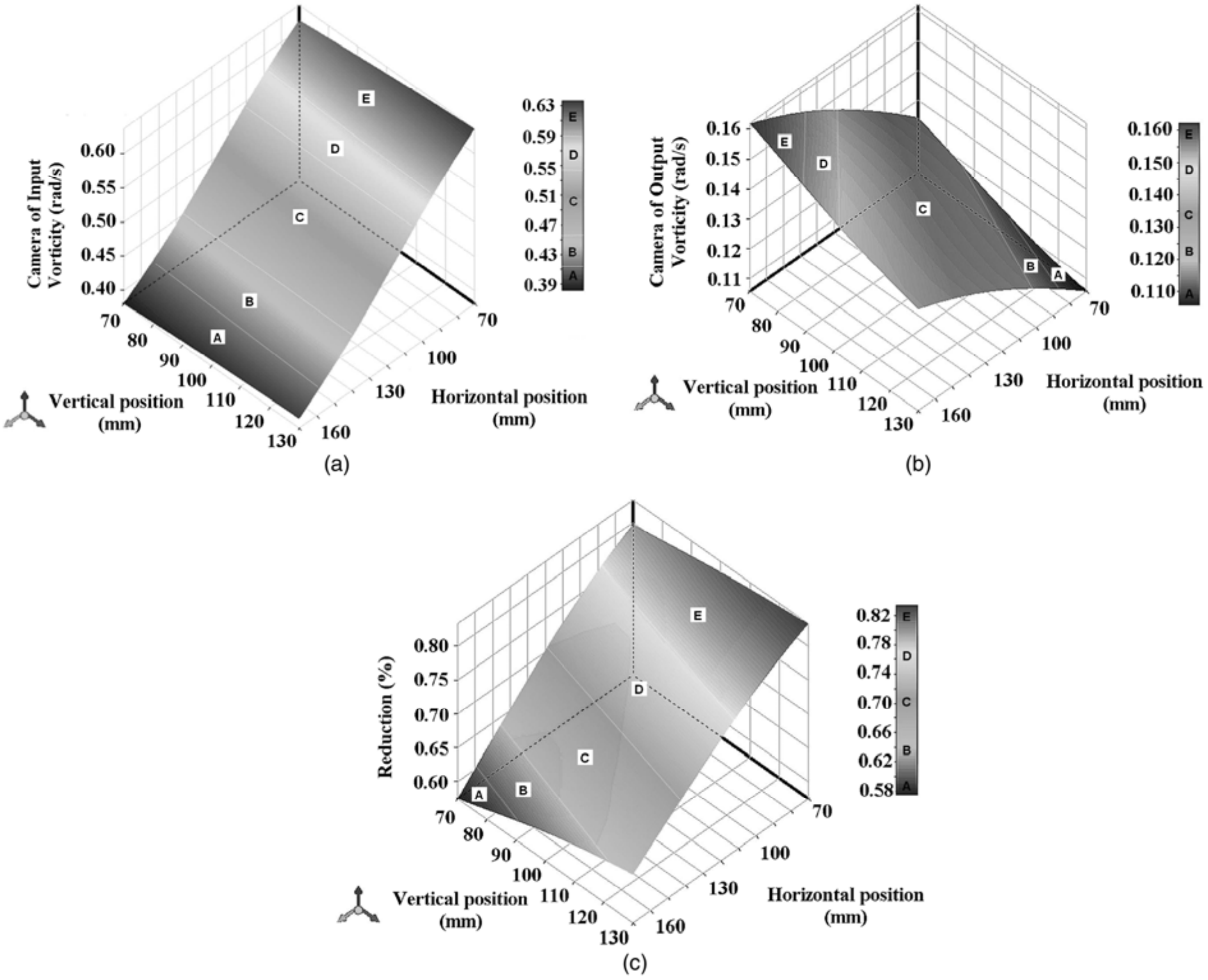


Figure 4. Response surface of the central panel position influence: a) average vorticity in the input chamber; b) average vorticity in the output chamber; c) vorticity reduction.

In Figure 4, it is shown that the horizontal displacement of the panel is the most influential input parameter in the vorticity results. On the one hand, the vertical movement of the panel does not change the vorticity values of the input chamber, as is shown in Figure 4(a). On the other hand, the vertical displacement in the output chamber is most important. In this case, if the panel is far from the bottom, the average vorticity in the output chamber will decrease, as it is shown in Figure 4(b). With respect to the horizontal displacement, response surfaces show that when the panel is moved away from the flow inlet, the vorticity decreases in the input chamber, whilst it increases in the output chamber, see Figures 4(a) and 4(b).

From the efficiency point of view, there is a maximum in the surface response which provides the best position for the panel. In this configuration, the value of the SCPT efficiency will be more than 80%, see Figure 4(c). From this maximum, the best horizontal and vertical positions for the panel are obtained: the best horizontal position is the nearest to the fluid inlet, about 70 mm from this; the best vertical position is when the flat panel is inserted into the SCPT as much as possible, which is about 130 mm between the end of the panel and the bottom of the SCPT.

Numerical and experimental comparison

Finally, the numerical simulation results and the laboratory tests results have been compared.

A comparison of these results has been carried out in two ways: a qualitative comparison of the fluid behaviour inside the SCPT and a quantitative comparison between the vorticity trends inside the SCPT.

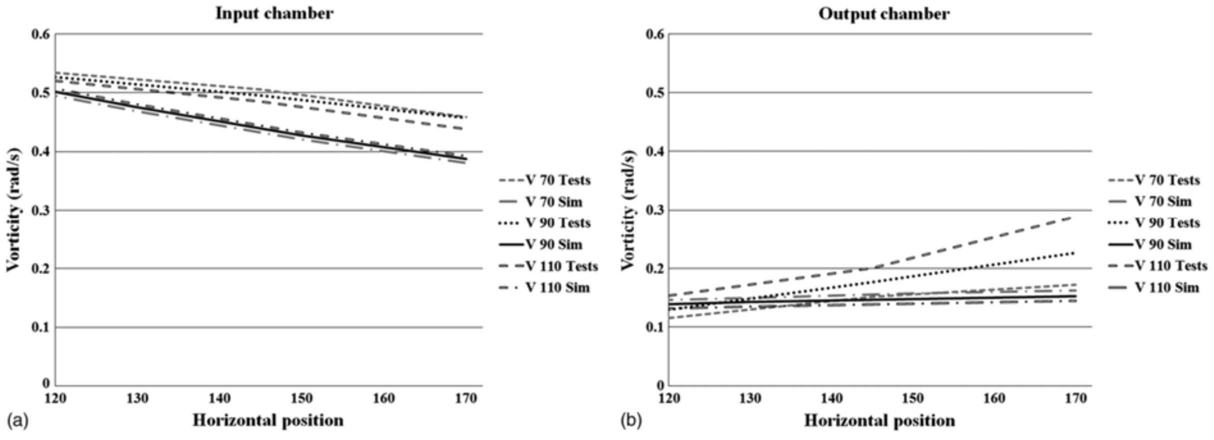


Figure 5. A comparison of vorticity results between the laboratory test and numerical simulation for: (a) the input chamber; (b) the output chamber.

Figure 5 shows the vorticity for three different vertical positions of the panel. Figure 5 shows the panel at a distance of 70 mm. at 90 mm. and finally, at 110 mm. from the bottom. Figure 5(a) shows the results of the average vorticity in the input chamber, and Figure 5(b), in the output one. All of these graphics show the vorticity trend with respect to the horizontal position, which is downward in the input chamber and upward in the output chamber. Furthermore, it is possible to see the high degree of correlation between the experimental and the numerical results, less than 15 % in mostly of the values. The variations among them are not very important due to the slight differences between the values and the similarity with trends in all positions studied.

In this sense, the results indicate that the laboratory tests cannot be completely reproduced due to the influence of the external factors. However, the vorticity trends obtained from the laboratory tests and the numerical simulations are in good agreement.

Finally, from the optimization of results obtained by DOE, it is possible to select the best panel position in order to achieve maximum efficiency. This optimum position is when the panel is near the fluid inlet and near the bottom of the SCPT. Comparing numerical and experimental results, it is observed that the optimum position is the same in both cases, see Figure 2a) H2-V1 position and the vorticity reduction surface response in Figure 4c.

CONCLUSIONS

In this research work the HE has allowed the optimization of the SCPT's behaviour taking into account experimental and numerical studies. In this sense, the efficiency of this system has been improved by means of the horizontal.

and vertical displacement of the panel. Results reveal the following main findings:

- A scale model was necessary to conduct the laboratory tests with the PIV equipment. Moreover, the PIV equipment requires a small fluid velocity variation; therefore the

SCPT scale model was used to reproduce the fluid behaviour with low velocity and less variation between the inlet and the outlet of the system.

- The laboratory tests indicate circular fluid movement inside the SCPT independent of the flat panel position. The fluid moves in two vortexes in opposite directions.

- In the input chamber, rotational velocity of the fluid is high. The majority of the particles are moving into the input chamber with the fluid. The velocity in the output chamber decreases because the volume of the output chamber is bigger. The particles in the output chamber precipitate at the bottom of the SCPT due to the low velocity in this volume.

- The horizontal displacement of the flat panel has more influence on the SCPT's performance than the vertical displacement. The horizontal displacement directly affects the velocity inside the input chamber. Furthermore, the horizontal displacement of the flat panel provides a large velocity differential between the input and the output chambers. In this way, an increase of the output 395 chamber volume is possible to allow the solid particles to precipitate.

- The best position for the flat panel is the nearest to the fluid inlet and the nearest to the bottom of the system. This position has been obtained in both laboratory tests and numerical simulations. All studies have been carried out for a constant flux, so the best position is limited to the fluid volume that goes into the SCPT. In extreme situations, like torrential rains, the best flat panel position could change in order to evacuate as much fluid as could be necessary.

- A good agreement was obtained between the techniques used, experimental tests and numerical simulations.
- The present study has proved that the design of experiments (DOE) in combination with finite volume modelling (FVM) could efficiently be used to optimize the best panel location in order to maximize the SCPT performance, decreasing the number of laboratory tests.

In summary, for new sustainable urban drainage systems (SUDS) it is a great advantage to be able to optimize and study the fluid behaviour of the SCPT using numerical models, with respect to an ecological design, which fulfils all serviceability requirements. These numerical simulations could be used in future studies to study the influence on the SCPT performance of different solid particles inside the fluid, the variation of the type of fluid or the inlet geometry, and so on.

Acknowledgements

The authors wish to express their gratitude to the Spanish Ministry of Economy and Competitiveness for the research project BIA2009-08272 funding. Furthermore, the authors greatly appreciate the collaboration of JA. Llaneza (University of Oviedo), John Bomidi and Professor Sadeghi (Purdue University). Besides, we also thank Swanson Analysis Inc. for the use of the ANSYS University Research program. Finally, we would like to acknowledge to the anonymous referees for their useful comments and suggestions.

REFERENCES

Caltrans. Treatment BMP technology report. California. Caltrans. 2010

Campbell, N.A.; D'Arcy, B.; Frost, A.; Novotny, V.; Sansom, A. 2004. Diffuse Pollution: An Introduction to the Problems and Solutions; IWA Publishing: London, UK.

Castro Fresno, D., Bayón, J. R., Rodríguez, J., y Ballester, F. 2005. Sustainable Urban Drainage Systems (SUDS). 30-5, 255-260.

Castro Fresno D., Rodriguez Hernández J., Del Coz Diaz J.J, Fernández Barrera A.H. inventors; Impulso Industrial Alternativo Ltd, COPROSA Ltd. Assignees, 2010. A new System to Collect, Pre-treat and Treat runoff water. Spanish Patent ES234619 (B1)

Castro-Fresno D., Andrés-Valeri V.C., Sañudo-Fontaneda L.A., Rodríguez-Hernández J. 2013. Sustainable Drainage Practices in Spain, Specially Focused on Pervious Pavements. Water, 5; 67-93

del Coz Díaz J.J., García Nieto P.J., Ordieres Meré J., Bello García A. 2007. Computer simulation of the laminar nozzle flow of a non-Newtonian fluid in a rubber extrusion process by the finite volume method and experimental comparison. J. Non-Cryst. Solids, 353 (8-10); 981–983

del Coz Díaz J.J., García Nieto P.J., Castro-Fresno D., Menéndez Rodríguez P. 2011. Steady state numerical simulation of the particle collection efficiency of a new urban sustainable gravity settler using design of experiments by FVM. Appl. Math. Comput. 217 (21), 8166–8178.

del Coz Díaz J.J., García Nieto P.J., Lozano Martínez-Luengas A., Suarez Domínguez F.J., Domínguez Hernández J. 2013. Non-linear numerical analysis of plywood board timber connections by DOE-FEM and full-scale experimental validation. Eng Struct 49, 76-90.

Dolz, J.; Gómez, M. 1994. Problems of stormwater drainage in urban areas and about the hydraulic study of collector networks [in Spanish]. Dren. Urbano, 1, 55–66.

Etalon Research Ltd. and Armfield Ltd. Prodel. 2009. H41 Laser PIV System manual for rtcontrol v 1.3, London.

Fernández Barrera A.H. 2009. Doctoral Thesis. Development of a water treatment system for runoff water from impervious parking surfaces using upflow and geotextiles. University of Cantabria.

Fernández-Barrera A.H., Rodriguez-Hernández J., Castro-Fresno D., Vega-Zamanillo A. 2010. Laboratory analysis of a system for catchment, pre-treatment and treatment (SCPT) of runoff from impervious pavements. Water Science and Technology. 61.7; 1845-52.

Flamant O., Cockx A., Guimet V.,Doquang Z. 2004. Experimental analysis and simulation of Settling process. Process safety and environmental protection, 82(B4); 312-318.

García Nieto P.J., del Coz Díaz J.J., Castro-Fresno D., Ballester Muñoz F. 2010. Numerical simulation of the performance of a snow fence with airfoil snow plates by FVM. Journal of Computational and Applied Mathematics, 234(4); 1200-1210.

Madenci E., Guven I. 2007. The Finite Element Method and Applications in Engineering Using ANSYS®. Springer. New York.

Mayurkumar S.G, Jyeshtharaj B.J, Pallippattu K.V. 2012. Study of two phase thermal stratification in cylindrical vessels: CFD simulations and PIV measurements. Chemical Engineering Science, 98; 125-151.

Melling A. 1997. Tracer particles and seeding for particle image velocimetry. Meas. Sci. Technol.,8, 1406.

Montgomery D.C., Design and Analysis of Engineering Experiments, 5th ed., John Wiley & Sons, New York, 2001.

Nezu I., Nakagawa H. 1993. Turbulence in open-channel flows. Balkema Publishers. Rotterdam.

Novotny, V. 2003. Water Quality: Diffuse Pollution and Watershed Management, 2nd ed.; Wiley: New York, NY, USA.

Penko A.M., Calantoni J. Three-dimensonal spatial variations of suspended sediment concentration over vortex ripples. Proceedings of Marine and River Dune Dynamics MARID IV. 2013. Bruges, Belgium.

Raffel M., et al. 1998. Particle image velocimetry: a practical guide. Springer Verlag.

Rodríguez-Hernández J., Fernández-Barrera A.H., Castro-Fresno D., Vega-Zamanillo A. 2010. Long-Term Simulation of a System for Catchment, Pretreatment, and Treatment of Polluted Runoff Water. J. of Environmental Engineering. 1442-1446.

Schroeder A., et al., 2008. Particle image velocimetry: new developments and recent application. Springer Verlag.

Van Hooff T., Blocken B., Defraeye T., Carmeliet J., van Heijst G.J.F. 2012. PIV measurements and analysis of transitional flow in a reduced-scale model: Ventilation by a free plane jet with Coanda effect. J. of Building and Environment, 56; 301-313.

Wang C-P, Sadeghi F., Wereley S.T., Chuang H.S. 2009. Investigation of fluid flow out of a microcavity using µ PIV. Tribol. Trans. 52, 817–832.

Wereley S. and Gui L. 2001. PIV measurement in a four-roll-mill flow with a central difference image correction (CDIC) method. International Symposium on Particle Image Velocimetry. Germany.

Zienkiewicz O.C., Taylor R.L., Nithiarasu P. 2005. The Finite Element Method for Fluid Dynamics. Butterworth-Heinemann. New York.