XXXVI Ibero-Latin American Congress on Computational Methods in Engineering
CILAMCE
Rio de Janeiro, 22-25 Nov 2015

EFFECT OF MORPHOLOGICAL FORM IN SEDIMENT ACCUMULATION IN 2D OBSTACLE

Carlos D. Mendez
Email: carlmendezg@gmail.com
Facultad de Ciencias Químicas UNA, Campus de la UNA, San Lorenzo, Central, Paraguay.

Christian E. Schaerer
Email: cschaer@pol.una.py
Polytechnic School, National University of Asuncion, Campus de la UNA, CP: 2160, P.O.Box: 2111 SL, San Lorenzo, Central, Paraguay.

Abstract. *In this work, the effects of the obstacle shape in the accumulation of sediments in a 2D system is investigated for several obstacles shapes. To this end, a two- dimensional numerical simulation with simultaneous fluid flow and scalar transport model are used. The different forms of the obstacles are subjected to the same flow conditions in order to detect the scalar accumulation front and behind the object. To solve the non-stationary incompressible Navier-Stokes equation and create the unstructured mesh we use the open source CFD software package OpenFoam and GMSH. The sediment transport is represented by a scalar transport equation, which is justified by the interest to determine accumulation zones for a great quantity of shapes.*

Keywords: *Scalar transport, sediment accumulation, Shape influence*

1 INTRODUCTION

The water erosion is a natural phenomenon which is part of the environmental processes. Due to its significant role in environmental processes, dragging large amounts of sediment due to water flows (rivers, waves and strong currents formed by the rains) is subject of many scientific investigations. The study of sediment transport due to water flows have been of great scientific interest because of their negative effects on the environment, mainly in watersheds and water reservoirs. This is due because nearly 1% of the total storage capacity World reservoirs is lost each year due to sediment transport (Mahmood 1987 and Yoon 1992) in addition to the reduced ability of river navigation in different parts of the world. Most of the models that have been developed in the past to provide information on the processes of erosion and water quality are inadequate to provide suitable for watershed scale, in addition to predictions based on events of the sediment load. They can suffer from a range of problems such as excessive parameterization unrealistic entry requirements, inadequate model assumptions or parameter values to local conditions and other problems of the models.

With increasing computing power in supercomputers in the last 20-30 years, there has been a rapid increase in exploration of the erosion and sediment transport through the use of models that are tested by computer. The models can be grouped into different groups according to their design; these are conceptual, from physical, empirical or union of previous models. Among the conceptual models we can find AGNPS (Young et al 1987), HSPF (Johanson et al 1980), EMSS (Watson et al 2001), LASCAM (Viney and Sivalapan 1999) and SWRRB (USEPA 1994); physical models are for instance ANSWER (Beasley et al 1980), CREAMS (Knisel 1980), GUEST (Yu et al 1997), LISEN (Takken et al 1999), PERFECT (Littlevoy et al 1992) and TOPOG (Haskins and Davey 1991) between others. These models are limited because of the many processes involved, especially the spatial distribution of variables. Even they have proposed models of sediment transport through vegetation (Lopez, 1998) and numerical models for sediment transport in open channels (Wu, 2000).

The reverse process to erosion is the deposition of the transported sediments. This settling process is affected by factors such as the hydrodynamic flow, which in turn, it is affected by the geometry of the channels or ducts wherein the fluid is flowing. Studies of the effect of geometry on different variables of interest in fluid mechanics such as the drag and lift of submerged bodies, the geometry of the channels, the turbulence and swirls, have laid the groundwork for the improvement in designing structures able to withstand the high flow rates of rivers for bridges or dams. Usually the main objective is to obtain a specific form to reduce the hydrodynamic drag.

The study of sediment transport processes is of interest to science and engineering because of their negative effects on the environment, mainly in watersheds and water reservoirs. Nearly 1% of the total storage capacity reservoirs in the world is lost every year due to sediment accumulations, in some cases the problem is the loss of valuable ecosystem due to entrainment of sediments. In addition the environmental degradation is strongly related to the transport of sediments (Mahmood, 1987). Normally the physical process of transport of sediments is studied either in laboratory flumes or using a specific hydraulic assembly. However, such experiments are time-intensive, costly and not always easy to conduct. In this context, the numerical simulation can reduce costs and improve the understanding of the relevant flow and transport phenomena.

There is a strong and unknown relations between the morphology of an obstacle and the sediment accumulation around it (Wu, 2000). This relation is influenced by the flow and the characteristic of the sediment. Numerous studies have focused on determining the effect of the morphology of the rivers in the deposition of sediments transported (Batalla (2003), Church (1995) and (Kantoush, *et al* 2009) . These hydrological studies focus on assessing how the rivers and their possible effects on the regions. Fewer works focus on correlating (Burkow, 20140) for optimizing the effect of the forms of the obstacles in the accumulation of sediment deposition.

This work is part of a serie of studies looking to find a method of optimization of forms of obstacles to allow the sediment accumulation to be reused in the recovery of areas of interest that are affected by hydraulic erosion. We consider the sediment transported as a passive scalar, therefore the size effect on flow patterns is not considered, future work will include such effect. We begin our analysis observing the possible accumulation regions formed by the presence of swirls. In this context, the main objective of this work consists in determining the presence of accumulation zones for different obstacles shapes under the same inflow velocities.

2 GOVERNING EQUATIONS AND NUMERICAL METHODS

Let $\Omega \subset \mathbb{R}^2$ be the spatial domain and Γ denote the boundary of Ω. We consider the following velocity-pressure formulation of the Navier-Stokes equations for governing the incompresible flow:

$$\nabla \cdot \mathbf{u} = 0, \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u}\mathbf{u}) = \nu \nabla^2 \mathbf{u} - \nabla p, \tag{2}$$

where $\mathbf{u}$ is a velocity field, p the pressure, ν the cinematic viscosity of fluid. The solution of this couple of equations is not straightforward because is not available an explicit equation for the pressure, One of the most common approaches is to derive an equation for the pressure by taking the divergence of the momentum equation and by substituting it in the continuity equation.

The momentum equation can be re-written in a semi discretized form as follows:

$$\mathbf{u}_p = \frac{H(\mathbf{u})}{a_p} - \frac{\nabla p}{a_p}, \tag{3}$$

where

$$H(\mathbf{u}) = -\Sigma_n a_n \mathbf{u}_n + \frac{\mathbf{u}_n}{\Delta t} \tag{4}$$

The first term of $H(\mathbf{u})$ represents the matrix coefficients of the n neighbouring cells multiplied by their velocity, while the second part contains the unsteady term and all the sources except the pressure gradient.

The continuity equation is discretized as:

$$\nabla \cdot \mathbf{u} = \sum_f \mathbf{S} \cdot \mathbf{u}_f = 0, \tag{5}$$

where $\mathbf{S}$ is outward-pointing face area vector and $\mathbf{u}_f$ the velocity on the face.

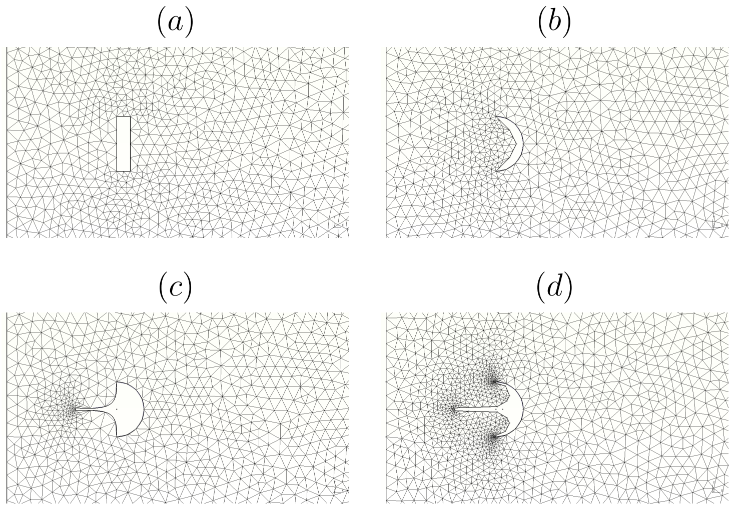

Figure 1: System analyzed with different obstacle shape and meshing: a) Front wall, b) Concave wall, c) Smooth arrow wall and d) Concave smooth arrow wall .

The velocity on the face is obtained by interpolating the semi discretized form of the momentum equation as follows:

$$\mathbf{u}_f = \left(\frac{H(\mathbf{u})}{a_p}\right)_f - \frac{(\nabla p)_f}{(a_p)_f}. \tag{6}$$

By substituting this equation into the discretized continuity equation obtained above, we obtain the pressure equation:

$$\nabla \cdot \left(\frac{1}{a_p}\nabla p\right) = \nabla \cdot \left(\frac{H(\mathbf{u})}{a_p}\right) = \sum_f \mathbf{S}\left(\frac{H(\mathbf{u})}{a_p}\right)_f. \tag{7}$$

Passive sediment transport. The scalar transport equation is presented bellow:

$$\frac{\partial c}{\partial t} + \nabla \cdot (\mathbf{u}c) \;\; = \;\; \nabla^2(D_c c), \tag{8}$$

where c is the concentration of sediments and D_c is the diffusion coefficient divided by the fluid density.

Boundary conditions. In order to perform the simulations we use the following boundary conditions: No-slip condition on the lateral wall and the object. Dirichlet condition: Velocity and scalar concentration for inflow. Neumann condition: Zero gradient for outflow and pressure in all domain. The effect of gravity is not considered.

3 THE NUMERICAL METHOD

The CFD simulations, the data pre- and post-treatment were performed using the OpenFOAM software toolbox (version 2.3.0) (OpenFOAM, 2015), which uses the finite volume approach with the spatial derivatives approximated by the first-order Gauss-linear scheme. The icoFoam solver was modified introducing the scalar transport equation and used for the Newtonian fluid flows simulations. Also the PISO algorithm was used to solve the velocity-pressure coupling.

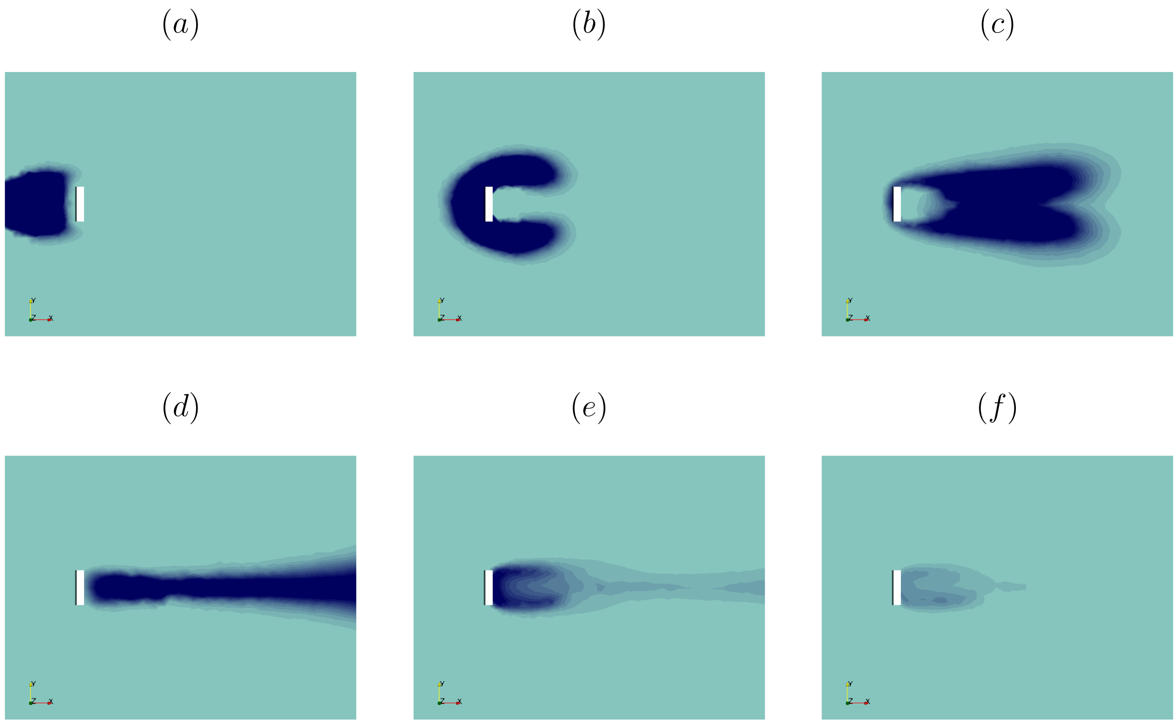

Figure 2: Simple wall simulations, results for different times in a top view : a) Time $0.5s$, b) Time $1.2s$, c) Time $2.5s$, d) Time $5.0s$, e) Time $10.0s$, f) Time $20.0s$.

Domain and meshing. The system consists in a 2D rectangular channel with 4 meters long and 3 meters wide where the income flow is lateral and head-impact with objects. The income scale is given with the flow in a region of size of 0.5 meters centered on the central axis, where the scalar c entered as unit step for a period of 0.6 seconds. The obstacles are placed at a distance of 1 meter from the edge, so as shown in the Figure 1.

Object shape selection. We study four different forms of obstacle presented in Figure 1, which are a simple wall, a curved wall, similar to a smooth shaft with a circular termination and a similar smooth termination arrow concave circular wall. Unstructured meshes were utilized with the open source software GMSH. With the GMSH grid generator is obtained a mesh of around 50.10^3 cells using the Delaunay algorithm.

Since it is intended to find possible accumulation scalar regions, (front, side and back area of the obstacle) four forms are proposed. A simple wall Figure1-(a) is used as a reference, then it seeks to increase sediment accumulation in front of the object, hence it is bent to have a curved concave wall Figure1-(b). This geometry reduce the fluid velocity and increase accumulation due to the low convective effect. To test the possibility of accumulating the scalar along the side of the wall, hence it is designed the third form 1-(c) being a smooth arrow wall that addresses the flow to the side.The fourth shape is looking for addressing the flow to areas of low speed combining the effect of directing the flow and low speed zones with the concave wall with the concave smooth arrow wall Figure1-(d).

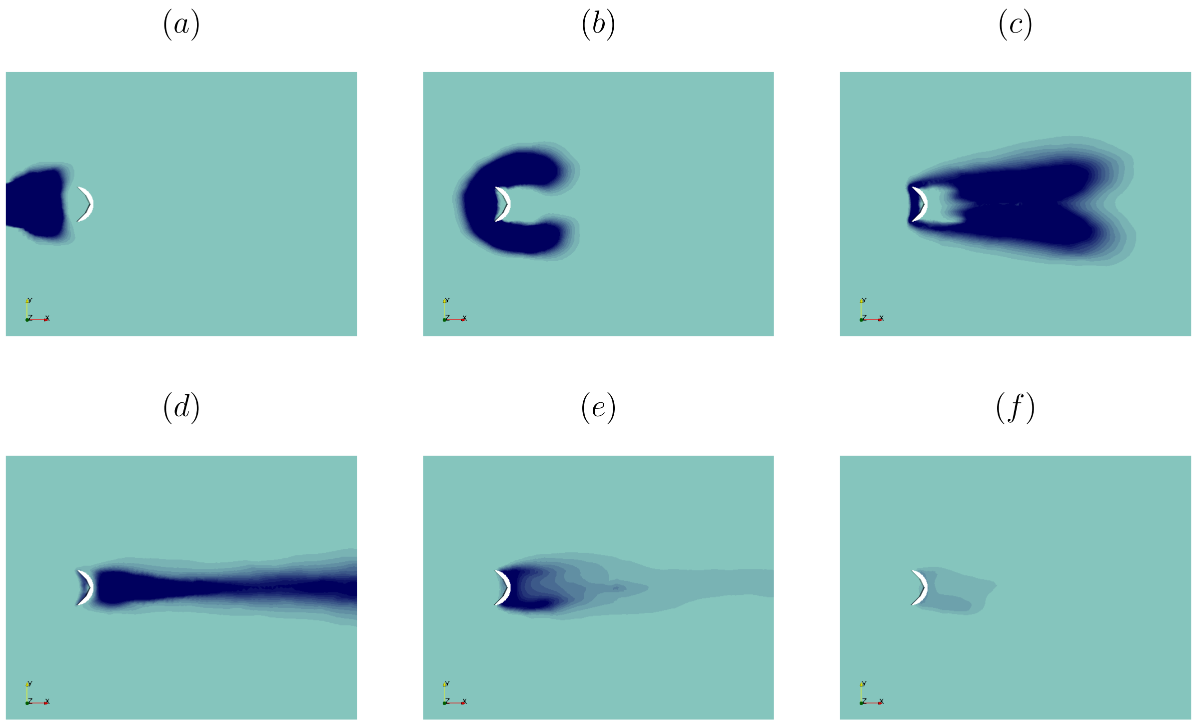

Figure 3: Concave wall test, results for different times in a top view : a) Time $0.5s$, b) Time $1.2s$, c) Time $2.5s$, d) Time $5.0s$, e) Time $10.0s$, f) Time $20.0s$.

4 RESULTS

This section presents experiments for evaluating the four shapes introduced before. We mainly analyze the front and back regions and the dwell-time of the mentioned regions. For the simulations we use the following transport properties: the cinematic viscosity ν is $0.01m^2/s$ and the diffusion coeficient D_c is $2.10^{-4}m^2/s$, and the Inflow velocity is $1m/s$.

Simple wall test. The first simulation present a simple wall as an object, the geometry used is presented in Figure1-(a) and the domain it's covered with 49.128 elements and 22.462 vertices. The total simulation time was $40s$ with $\delta t = 0.005s$. The results are presented in Figure 2 for six times. Observe two main aspects. The first is the low accumulation of the scalar in front of the wall. This is influenced due to the lack of a low velocity zones that allow the accumulation of scalar, thus the dwell time passive scalar is low (about 3 seconds). On the other hand, a second aspect is the accumulation of sediments achieved in the back of the wall. In this case, an area of low pressure keeps the sediment into a region equivalent to those formed by the breakdown of the boundary layer, therefore the dwell time of the scalar are high (the order of 35 seconds). This experiment gives an idea that these objects can be achieved residence times in the back of the object that would favor the deposition of sediments accumulation.

Concave wall test. The first simulation gives an idea to how the scalar passive it behaves. With this in mind we consider a concave wall. The geometry used is presented in Figure1-(b) and the domain is discretized with around 49.178 elements and 22.475 vertices. The total simulation time is $40s$ with $\delta t = 0.001s$.

The results are presented in Figure 3 for six times. The concave shape of the wall reduces the speed of fluid in the front, therefore part of the scalar is "stuck" in this region, the rest of

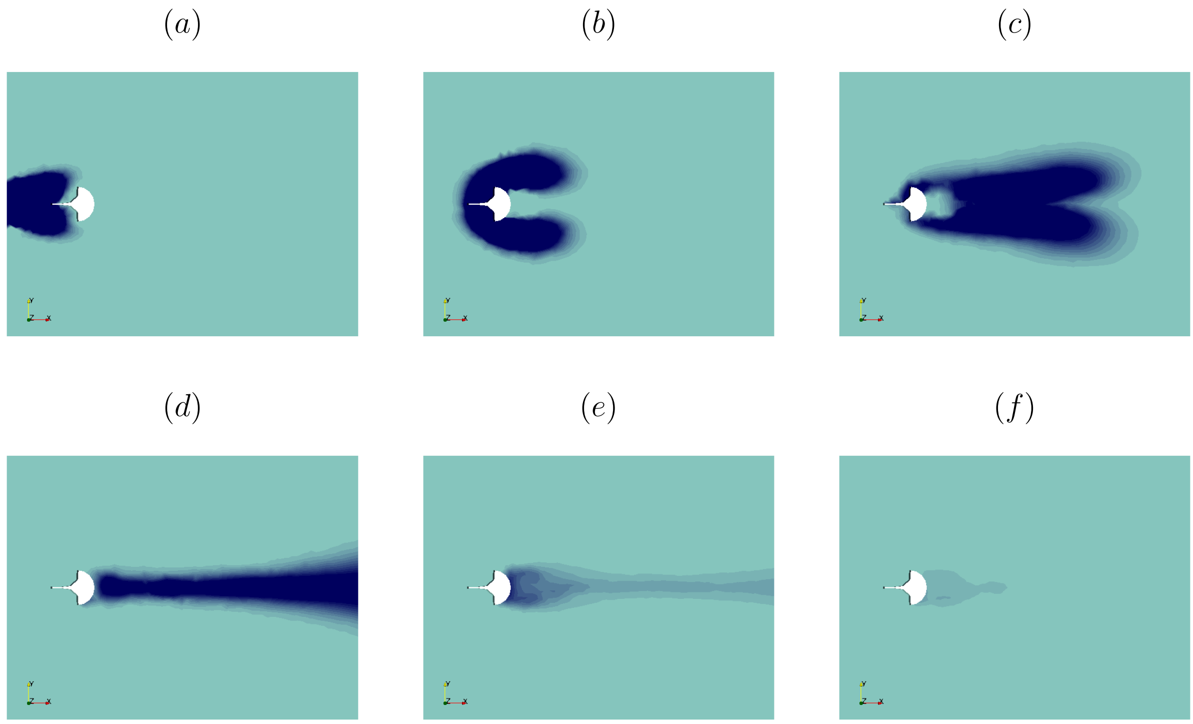

Figure 4: Smooth arrow wall test, results for different times in a top view : a) Time $0.5s$, b) Time $1.2s$, c) Time $2.5s$, d) Time $5.0s$, e) Time $10.0s$, f) Time $20.0s$.

the scalar is transported by the fluid, the dwell time in this region is higher (the order of 30.5 seconds). Accumulation in the back is lower than in the single wall due to the curved wall, which reduces the region break of the boundary layer. Therefore sediment is carried by the fluid stream, the dwell time the scalar is smaller than the wall (about 27.6 seconds).

Smooth arrow wall test.

The two previous cases have the capacity to accumulate the sediment on the front of the obstacle. It is observed only two regions of accumulation: the front region to the object and the back region of the object. With the aim of achieve low velocity zones on the lateral sides of the objects to accumulate the sediments in those regions, a smooth arrow on the front and circular in the back wall it is proposed. The geometry used is presented in Figure1-(c) and the domain it is discretized with 47.726 elements and 21.815 vertices. The total simulation time is $40s$ with $\delta t = 0.005s$.

Results are presented in Figure 4 for six times. It is observed that even trying to accumulate sediments to the lateral regions of the object, this does not happen. The accumulation of sediments on the front is low as the dwell time of the scalar (about 4.0 seconds), and in the back of the object was similar to the previous case due to its similarity, the dwell time was the same (about 27.6 seconds).

Concave smooth arrow wall test In this case seeking to increase the dwell time of sediments on the front of the object, we purpose a geometric shape that can direct the flow to areas of low speed. To this end it is used the geometry presented in Figure1-(d). The domain is discretized with 60.305 elements and 27.505 vertices. The total simulation time is $40s$ with $\delta t = 0.001s$.

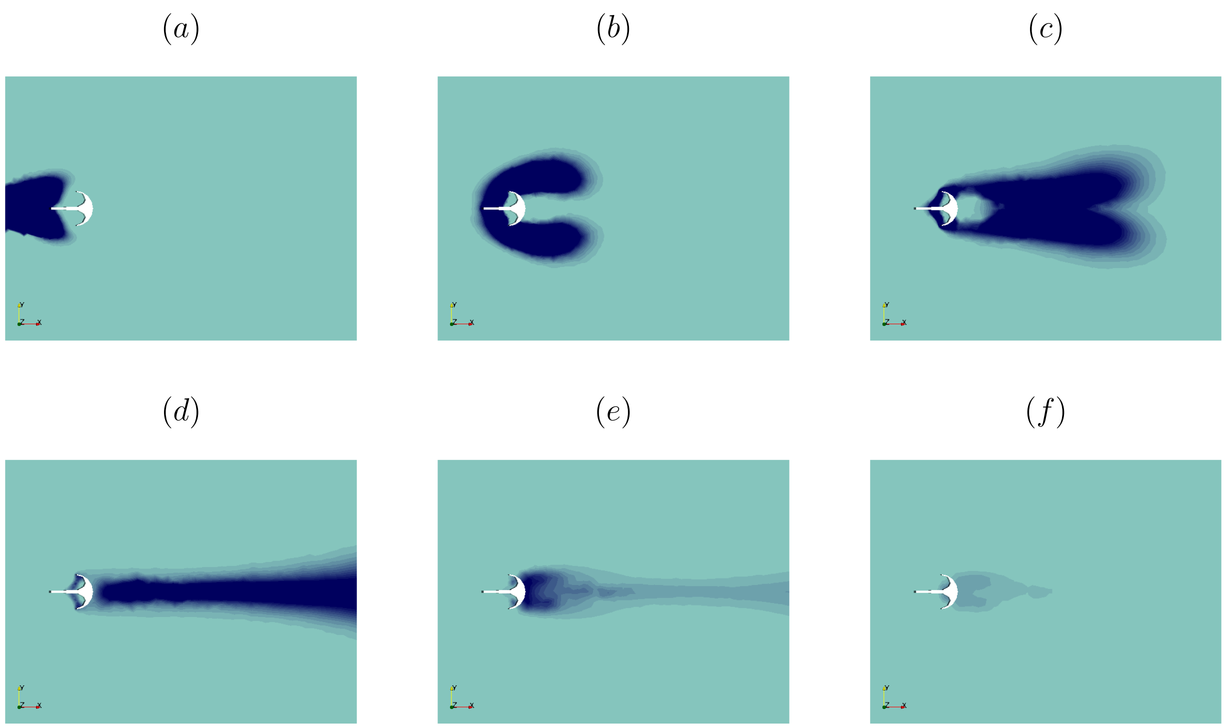

Figure 5: Concave smooth arrow wall test, results for different times in a top view : a) Time $0.5s$, b) Time $1.2s$, c) Time $2.5s$, d) Time $5.0s$, e) Time $10.0s$, f) Time $20.0s$.

The results are presented in Figure 5 for six times. With this object it is increased the dwell time (about 36 seconds) and the amount of sediments on the front of the object. This due to the reduction of speed with direction pointed to object. The behavior in the back of the object is similar to the previous cases and even the dwell time of the scalar was similar order (about 28 seconds).

5 CONCLUSIONS

Even with the simplicity of this study it is observed that two preferential accumulation of sediments regions are identified: the front and back region of the object. At the object front the accumulation it is favored by concave shapes that reduce the flow velocity, while in the back region, the accumulation is favored by the breakdown of the boundary layer. Concave shapes favor accumulation of sediments in the frontal region, while planar shapes perpendicular to the flow favor the accumulation and retention in the region back; the dwell time follow the same trend. At the moment, authors are working in algorithms for optimization and control of shapes.

ACKNOWLEDGEMENTS

CES acknowledges the PRONII - CONACyT- Paraguay. Authors acknowledge the support given by the NIDTEC - LCCA the Laboratory of Scientific and Applied Computing, Polytechnic School - UNA - Paraguay.

REFERENCES

Batalla, R.J., 2003. Sediment deficit in rivers caused by dams and instream gravel mining. A review with examples from NE Spain. *Cuaternario y Geomorfologa* 17 (34), 7991

Beasley, D.B., Huggins, L.F., Monke, E.J., 1980. ANSWERSa model for watershed planning. *Trans Am Soc Agric Eng 23*, 938944.

Burkow, M., Griebel M., 2014. *Full three dimensional numerical simulation of the sediment transport and the scouring at a rectangular obstacle*. PhD thesis, Instituf fur Numerische Simulation, University of Bonn.

Church, M., 1995. Geomorphic response to river flow regulation: case studies and time-scales. *Regulated Rivers: Research and Management* 11 (1), 322.

G. Haskins, 1991. Integrated Quantity/Quality Modelling-Stage 3, *Department of Water Resources*, Sydney, pp. 102.

Hanley, N., Faichney, R., Munro, A., Shortle, J.S., 1998. Economic and environmental modelling for pollution control in an estuary. *Journal of Environmental Management* 52, 211225.

Kantoush, S. A. and Schleiss, A. J. 2009. Channel Formation in Large Shallow Reservoirs with Different Geometries during Flushing, *Journal of Environmental Technology*, Volume 30 Issue 8, 855-863.

Knisel, W.G., 1980. CREAMS: A Field Scale Model for Chemicals, *Runoff and Erosion from Agricultural Management Systems*. USDA.

Mahmood, K., 1987. Reservoir sedimentation: IMPACT, Extent, and Mitigation. *World Bank Technical Paper Number 71*Washington, D.C.

Laflen, J.M., Lane, L.J., Foster, G.R., 1991. WEPP: A new generation of erosion prediction technology. *Journal of Soil and Water Conservation* 46, 3438.

Lpez, F. 1998. Open-channel flow through simulated vegetation:Suspended sediment transport modeling. *Water Resources Rsesearch*, Vol 34, NO. 9, Pages 2341-2352, September 1998.

OpenFOAM. *http://www.openfoam.com*. 2015.

R. Roth, S. Ulbrich, 2013. A Discrete Adjoint Approach for the Optimization of Unsteady Turbulent Flows. *Flow Turbulence Combust* Vol 90. Pages 763783.

Takken, I., Beuselinck, L., Nachtergaele, J., Govers, G., Poesen, J., Degraer, G., 1999. S*patial evaluation of a physically-based distrib- uted erosion model (LISEM)*. Catena 37 (3-4), 431447.

Wu, W., 2000. 3D Numerical modeling of flow and de sediment transport in open Channels, *Channels, Journal of Hydraulic Engineering*, Vol. 126, No. 1, January, 2000.

Yoon, Y.N., 1992. The state ande the perspective of the direct sediment renoval methods from reservoirs, *International Journal of sediment Research*, 7(2): 99-116.

Yu, B., Rose, C.W., Cielsiolka, C.A.A., Coughlan, K.J., Fentie, B., 1997. *Towards a framework for runoff and soil loss prediction using GUEST technology*. Australian Journal of Soil Research 35, 11911212.