Numerical modelling of bed sediment particle tracking in open channel with skewed box-culvert

Israel E. Herrera-Díaz*†, Mariano López-Amezcua††, Noe Saldaña-Robles† and Javier Zavala Sandoval†

† Depto. Ingeniería Agrícola, DICIVA, Campus Irapuato-Salamanca, Universidad de Guanajuato, México e-mail: eherrera@ugto.mx, saldanar@ugto.mx, web page: http://www.ugto.mx/campusirapuatosalamanca/

†† Programa de Ingeniería Civil, DCSI, Campus Celaya-Salvatierra, Universidad de Guanajuato, México, e-mail: m.lopezamezcua@ugto.mx, web page: http://www.ugto.mx/campus-celaya-salvatierra

ABSTRACT

A particle tracking model was applied to estimate the bed sediment transport in open channel with skewed box-culvert in rivers in Mexico, for which purpose the calculation of the hydrodynamics of the study channel was determined the three-dimensional velocity field [1], later, the calculation of particle transport was obtained, which was determined in any direction of the space caused by the velocity field and the turbulent dispersion (random movement of the Brownian type). The dispersion and re-suspension mechanisms of the particles used were represented by stochastic models, which describe the movement by means of a probability function [2]. The validation of the model was previously carried out by [3], obtaining average relative errors of less than 4.8%.

Three numerical scenarios were calculated including different alternatives and its behaviour at the entrance, interior and exit of the water flow in the construction to determine which is the best option to be used on the skewed multi barrel crossings. In order to accomplish this, a variable slope channel and 1: 60 scale models of box culverts with 10, 22 and 45 degrees of skewedness were used.

The results observed in the multi-eyed box culverts were favourable, due to the fact that the speed spans are low increase inside and outside of them, which favours the hydrodynamic behaviour and minimize the accumulation of sediment into structure in the river.

INTRODUCTION

The analysis and design of culvert for the flow of water through bridges, roads or road infrastructure works are important to minimize environmental problems and environmental impact on rivers and streams in the area of construction and civil engineering.

The behavior of these culverts has been studied in relation to the number and placement of these in the bridges or crossings over rivers and streams, all depending on the geographical, hydrological and construction characteristics, always with the purpose of minimizing the natural channel and associated environmental processes such as sediment transport.

For the study of these structures that facilitate the transit of water through roads and bridges, the use of numerical models is necessary to test different arrangements and configurations before the hydraulic scenarios that occur in the area at different times of the year.

Various results have been proposed by works related to the hydrodynamics of rivers, irrigation channels and control works to regularize fluid flow and flow measurement.

The numerical computational model developed in this work for the transport of the particles is governed by the Lagrangian approach, where the particles are located following a concentration exponential law or randomly located in the space. The advantage of using Lagrangian models to estimate sediment transport and some temporal changes in the morphology of the bottom lies in the computational speed of using a previously calculated hydrodynamic field for the movement of particles. These allow the approximation of the temporal concentration of sediments contemplating the density of the material and using the PIC method to quantify the sediment transport associated with the displacement of the particles near the bottom.

The Lagrangian approach is widely used in the study of the trajectories of movement of solid particles in fluid environments. This is due to the fact that it is possible to track the movement for each specific particle in more detail, in comparison with determining average concentration for grid cells. However, in practice, the most difficult part to use a Lagrangian based method is the strong dependence on the performance of computational resources, such as the amount of memory required for the particles.

The Lagrangian mathematical approximations, based on methods of random movement, are well established tools for the calculus of sediment transport and pollutant discharges into aquatic environments. The discharges are treated as a finite number of particles; these particles move under the influence of the previously established flow field.

The results of the hydrodynamic calculation correspond to a stable field and converged in time, which indicates that the velocity fields and their turbulent parameters can be assumed as constants, but with an important spatial variation, in this way the simulation of the transport of particles is performed for times greater than those obtained in the hydrodynamic simulation. Therefore, for the transport of particles, the same hydrodynamic field can be used repeatedly, for all time intervals, (Dt) as many times as required, until the simulation period is completed. The velocities of the particles are obtained by linearly interpolating the velocities around the particle in the three-dimensional mesh. An advantage of separating the hydrodynamic simulation from the simulation of particle movement, allows us to develop large numbers of transport simulations [4], in this way we can simulate particle movements with: different locations and types of sources, several simulation durations, different transport parameters and different physical properties of the particles (specific weight and diameter), all this based on a hydrodynamic velocity simulation.

According to some current review papers regarding the Lagrangian modelling of Saltating Sediment Transport [5, 6], a model for the transport of sediment has to include mainly the motion of saltating grains, diffusion of particles, and calculation of bedload transport rate and to improve the motion of particles representing more natural shapes. And, for rivers, it must consider the nonuniform character of sand, distribution of particle saltation lengths, and excursion lengths may be more important in determining the morphodynamic behavior of the channel bed than the average particle motion.

The application problem was to estimate the bed sediment transport in open channel with skewed box-culvert in rivers in Mexico, the study was in specific three different scenarios with a variable slope channel laboratory and 1:60 scale models of box culverts with 10, 22 and 45 degrees of skewedness were used, all conditions were controlled on flow rate and velocities to get the hydrodynamics field.

Three sediment particle diameters were considered in all domain, shape regular and sand material properties in all simulation time; the seeded of 700,000 particles is homogeneous throughout the domain, 3 different diameters D15, D50 and D90 are included with a percentage distribution of 40, 40 and 20 respectively of the total particles.

METODOLOGY

The governing equation hydrodynamical model

The Navier-Stokes equations in free surface flow, in cartesian coordinates; use the hypothesis of hydrostatic pressure and considering the postulates of Reynolds [7].

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-g\frac{\partial \eta}{\partial x}+div\left(\nu_e\overrightarrow{grad}(u)\right)$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=-g\frac{\partial \eta}{\partial x}+div\left(\nu_e\overrightarrow{grad}(v)\right)$$

$$\frac{\partial \eta}{\partial t}=-\frac{\partial}{\partial x}\left(\int_{-zf}^{\eta}u\,dz\right)-\frac{\partial}{\partial y}\left(\int_{-zf}^{\eta}v\,dz\right) \tag{1}$$

where $\boldsymbol{\nu_e}$ effective viscosity coefficient, obtained by adding the turbulent and molecular viscosity coefficient $\boldsymbol{\nu_e=\nu_t+\nu_m}$, [8] proposes the following model to solve the turbulent viscosity:

$$\nu_t=\left\{{\ell_h}^4\left[2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2\right]+{\ell_v}^4\left[\left(\frac{\partial u}{\partial z}\right)^2+\left(\frac{\partial v}{\partial z}\right)^2\right]\right\}^{\frac{1}{2}} \tag{2}$$

where the vertical length scale $\boldsymbol{\ell_v=\kappa(z-z_b)}$ for $\frac{(z-z_b)}{\delta}<\frac{\lambda}{\kappa}$ and $\boldsymbol{\ell_v=\lambda\delta}$ for, $\frac{\lambda}{\kappa}<\frac{(z-z_b)}{\delta}<\mathbf{1}$ $\boldsymbol{\kappa}$ is the von Kármán constant typically 0.41, $(\boldsymbol{z-z_b})$ is the distance from the wall, $\boldsymbol{\delta}$ is the boundary-layer thickness and $\boldsymbol{\lambda}$ is a constant, typically 0.09. In the case of shallow-water flows, due to a steady current, the boundary-layer thickness may be assumed to be equal to the water depth h. The horizontal length scale is usually different than the vertical length scale, and the simplest assumption is to assume direct proportionality defined by $\boldsymbol{\ell_h=\beta\ell_v}$. The constant $\boldsymbol{\beta}$ has to be determined experimentally.

Free surface and bottom conditions

$$\tau_x^{fondo}=\nu_e\frac{\partial u}{\partial z}\bigg|_{fondo}=\frac{g\sqrt{u^2+v^2}}{Cz^2}\,(u)$$

$$\tau_y^{fondo}=\nu_e\frac{\partial v}{\partial z}\bigg|_{fondo}=\frac{g\sqrt{u^2+v^2}}{Cz^2}\,(v) \tag{3}$$

where $\boldsymbol{Cz}$ is the Chezy friction coefficient. The velocity components are taken from values of the layer adjacent to the sediment-water interface.

$$\boldsymbol{\tau_x^{sup}=\nu_e\frac{\partial u}{\partial z}\bigg|_{superficie}=-\frac{\rho_{aire}}{\rho_{agua}}a_{viento}\omega_x|\omega_x|}$$

$$\tau_y^{sup}=\nu_e\frac{\partial v}{\partial z}\bigg|_{superficie}=-\frac{\rho_{aire}}{\rho_{agua}}a_{viento}\omega_y|\omega_y| \tag{4}$$

where $\boldsymbol{\rho_{aire}}=1.29$ kg/m3, $\boldsymbol{\omega_x}$ y $\boldsymbol{\omega_y}$ are the horizontal components at x and y respectively of the wind speed at 10 m altitude. The unidimensional coefficient $\boldsymbol{a_{viento}}$ can be obtained using the equation given by [9].

$$a_{viento} = 0.565 \times 10^{-3}; \;\; si\ |\vec{\omega}| \leq 5\ m/s$$

$$a_{viento} = (-0.12 + 0.137|\vec{\omega}|)10^{-3}\ ; \quad si\ 5 \leq |\vec{\omega}| \leq 19.22\ m/s$$

$$a_{viento} = 2.513 \times 10^{-3}; \;\; si\ |\vec{\omega}| \geq 19.22\ m/s \tag{5}$$

The governing equation particle tracking model

The numerical model for particle transport is given under a Lagrangian approach; the particles are placed following an exponential law of concentrations or by an initial position in three-dimensional space [10]. For the movement of particles, a stochastic model is considered and discretized in three dimensions (Fig. 1), considering the specific weight of each particle as well as the fall velocity of the same [11] and it is verified if these are within the domain study for a single time step (Dt) from (n) to (n + 1) is given by:

$$x_i^{n+1} = x_i^n + u_{i,j,k}(\Delta t) \pm (2rand(iseed) - 0.5)\sqrt{(2vt_{i,j,k}\Delta t)}$$
$$y_i^{n+1} = y_i^n + v_{i,j,k}(\Delta t) \pm (2rand(iseed) - 0.5)\sqrt{(2vt_{i,j,k}\Delta t)}$$
$$z_i^{n+1} = z_i^n + w_{i,j,k}(\Delta t) \pm (2rand(iseed) - 0.5)\sqrt{(2vt_{i,j,k}\Delta t - w_s\Delta t)} \tag{6}$$

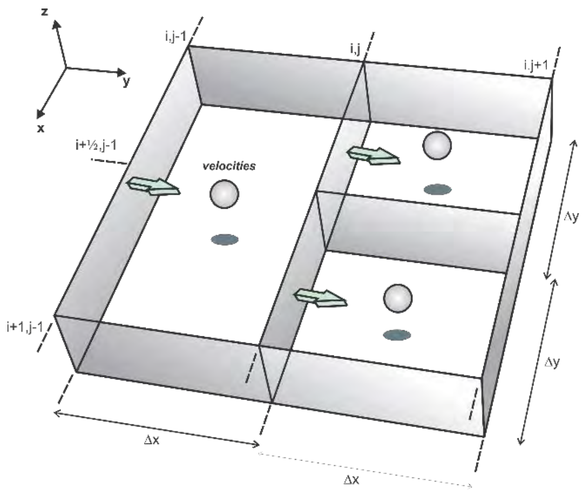


Fig. 1. Location in the three-dimensional space of the particle and its associated velocities

where (x_i^n, y_i^n, z_i^n) is the position of the particle in time (t), (u, v, w) are the average velocities in (i, j, k), (ν_t) is the turbulent viscosity coefficient, (Dt) is the Lagrangian time step and (w_s) is the velocity of sediment falling. The tracking of the particles is using the Eq. (6), therefore each particle is subject to a spatial displacement of magnitude $\pm(2rand(iseed) - 0.5)\sqrt{(2vt_{i,j,k}\Delta t)}$, in any direction of the domain, the sign is positive or negative depending on the sense of its location. In that moment of time the velocity field acts on each particle, in this way the movement has a sense in function of the main movement, given by the velocity fields. The term (ν_t) is found over the entire domain, represented by a field of positive scalars, which possesses information of turbulent intensities.

The shear forces in a turbulent flow, along its depth (z), can be written as:

$$\tau_i = \rho\nu\frac{dU}{dz} - \rho\overline{u_\iota w} \tag{7}$$

where: (τ_i) bottom shear stress, (ρ) fluid density, (υ) cinematic viscosity coefficient, (U) mean fluid velocity and $(\overline{u_\iota w})$ bottom double correlation.

The critical shear stress that perform on the particles, is written:

$$\tau_{critico} = 0.03(\rho_s - \rho)gd_{50} \qquad (8)$$

where: (ρ_s) solid density, (ρ) water density, (g) gravitational constant and (d50) particle diameter 50%.

The probability function for the deposition of the particles is determined with the following equation:

$$P_{depósito} = \left\{ \begin{matrix} 0 \\ \left(1 - \frac{\tau_{x,y}}{\tau_{crítico}}\right) \end{matrix} \right\}, \begin{matrix} \tau_{x,y} \geq crítico \\ \tau_{x,y} < crítico \end{matrix} \qquad (9)$$

And the probability function for the resuspension of the particles is established in the following way:

$$P_{depósito} = \left\{ \begin{matrix} 0 \\ \left(1 - \frac{\tau_{crítico}}{\tau_{x,y}}\right) \end{matrix} \right\}, \begin{matrix} \tau_{x,y} \leq crítico \\ \tau_{x,y} > crítico \end{matrix} \qquad (10)$$

Velocity fall of sediment

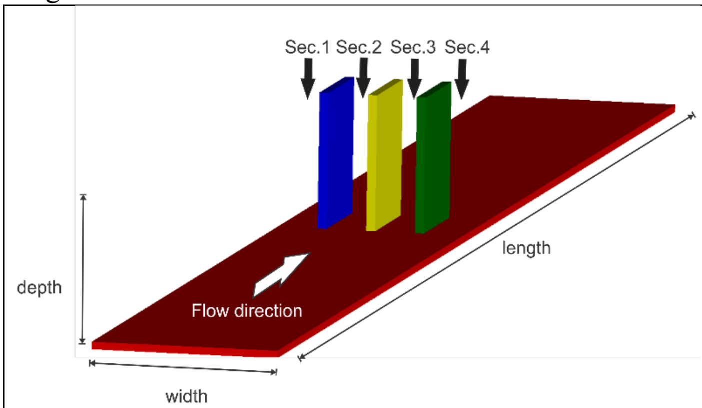
To determine the velocity fall of sediment particles, these are considered to have non-spherical shape, so the effect of the shapes have a considerable influence on their velocity, mainly on relatively large particles (>300µm), the expressions that determine the magnitude of velocity fall [12] are expressed below.

$$\begin{aligned} w_s &= \frac{(S-1)gd^2}{0.8\nu}; \;\; 1 < d \leq 100\mu m \\ w_s &= \frac{10\nu}{d}\left[\left(1 + 0.01\frac{(S-1)gd^3}{\nu^2}\right)^{0.5} - 1\right]; 100 < d \leq 1000\mu m \\ w_s &= 1.1\left((S-1)gd^2\right)^{0.5}; \;\; d > 1000\mu m \end{aligned} \qquad (11)$$

where (d) diameter of particle, (S) specific gravity, (υ) kinematic viscosity coefficient y (g) gravitational constant.

RESULTS AND APPLICATION

The physical scale open channel (Fig.2) had 2.32 m length, 0.18 m depth and 0.25 m width; the flow rate was 0.0019 m3/s, slope 0.006 and three skewed multi barrel crossings with 0, 10, 22 and 45 degrees.

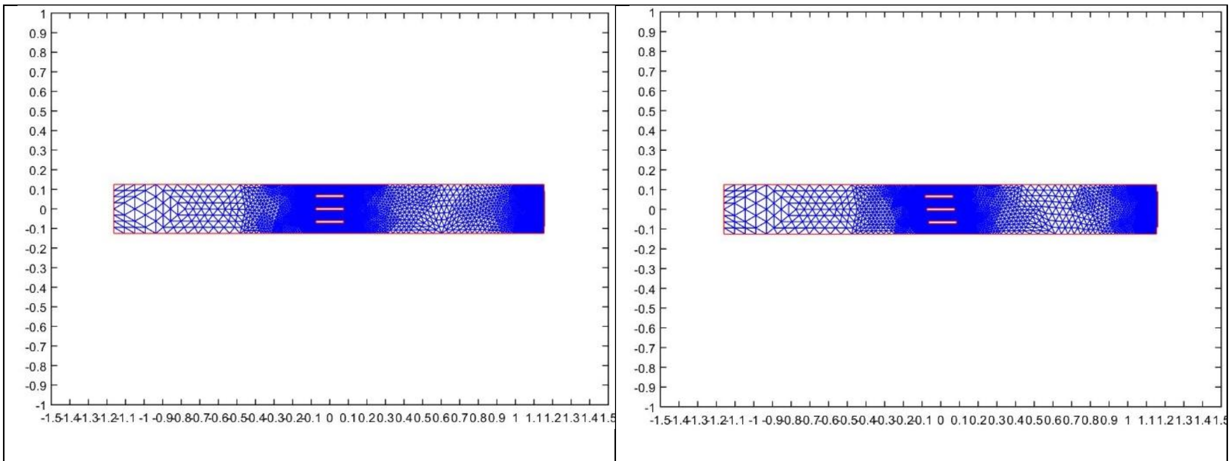


Open channel scheme

Open channel laboratory

Fig. 2. Open channel domain

The numerical model was worked with a mesh in finite elements (Fig. 3) the first scenario contained 8986 nodes and 17280 triangles to zero degrees of skewedness; the second scenario contained 7730 nodes and 14816 triangles to ten degrees of skewedness; the third scenario was 2034 nodes and 3736 triangles to twenty-two degrees of skewedness; and the last scenario was 8178 nodes and 15712 triangles to forty-five degrees of skewedness were simulated with increments of Dt = 0.01 s and total time of 1500 s.



Scenario 1: zero degrees of skewedness

Scenario 2: ten degrees of skewedness

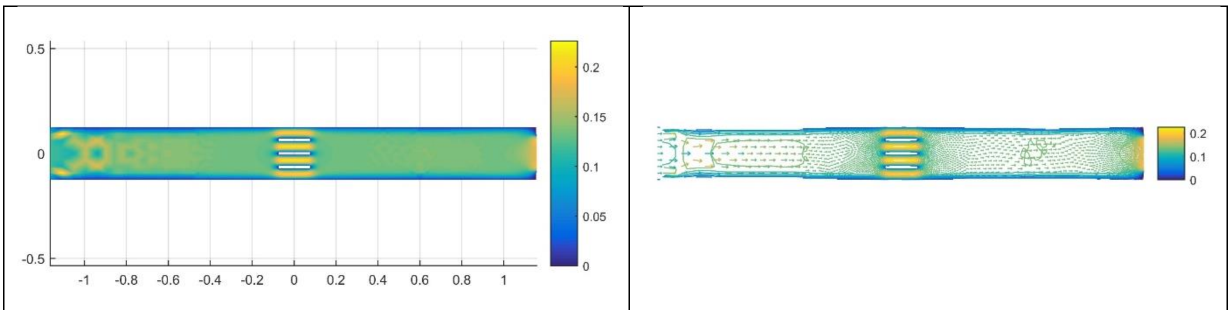
Scenario 3: twenty-two degrees of skewedness

Scenario 4: forty-five degrees of skewedness

Fig. 3. Finite element mesh in different degrees of skewedness

The hydrodynamic model was used to generate the velocity field corresponding to 1500 s., taking as force the magnitude of flow rate in all scenarios.

In the Fig. (4 to 7) the results of the behavior of the hydrodynamic in laboratory channel are presented, it is observed that the velocity magnitude had change in each scenario, especially near to the box-culvert.



Velocity magnitude (m/s)

Field velocity vector (m/s)

Fig. 4. Hydrodynamic field in zero degrees of skewedness (m/s)

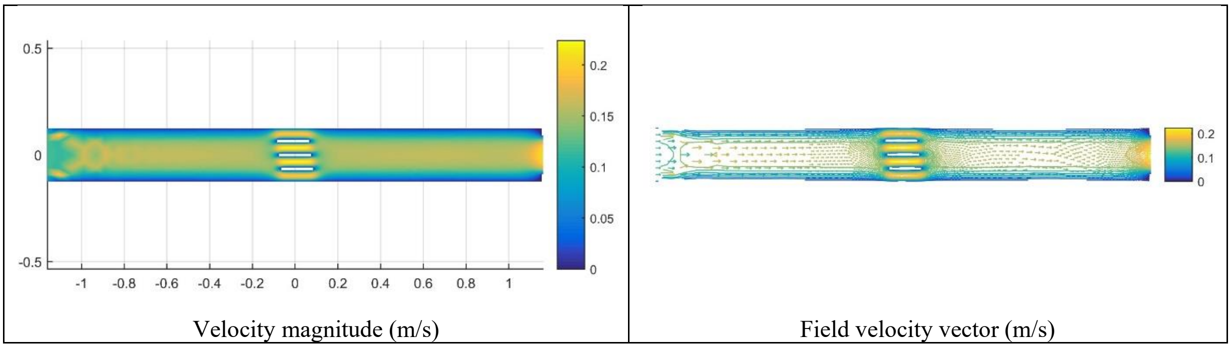


Fig. 5. Hydrodynamic field in ten degrees of skewedness (m/s)

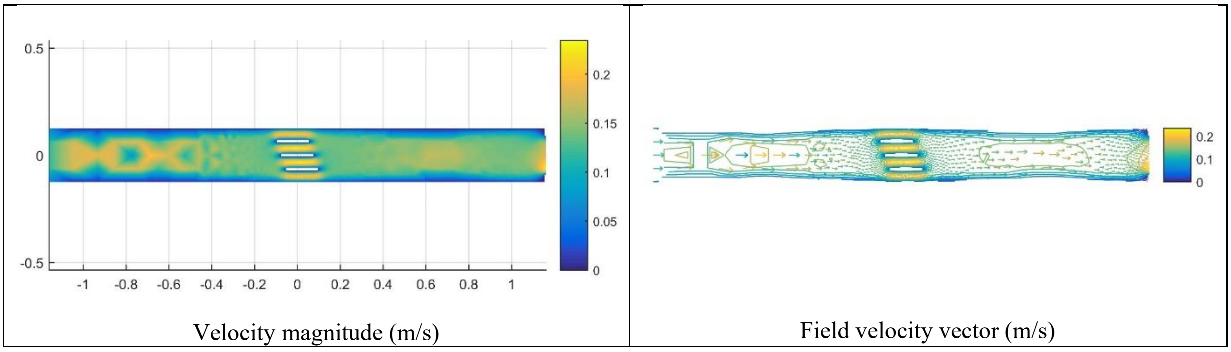


Fig. 6. Hydrodynamic field in twenty-two degrees of skewedness (m/s)

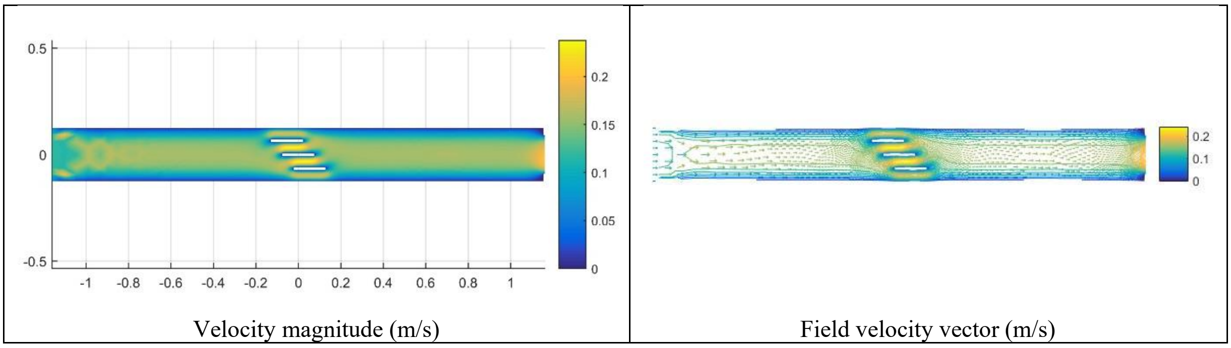


Fig. 7. Hydrodynamic field in forty-five degrees of skewedness (m/s)

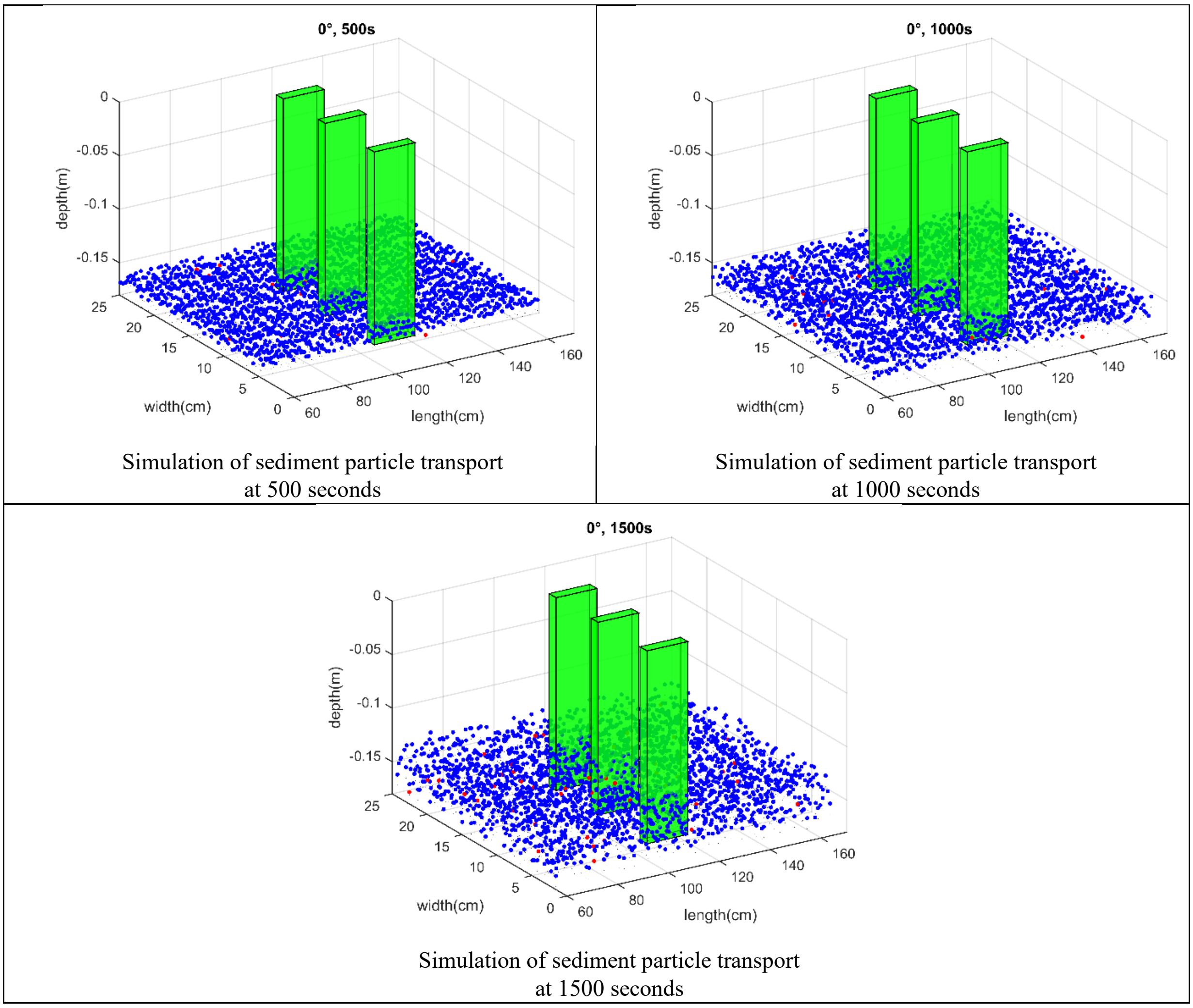
Once the velocity field has been obtained, the seeding of sediment particles is placed at the bottom, the characteristic diameters of the sediment in the study area, which feed the sediment model are mentioned in the table (1), the material is constituted in 87% sand and the remaining 13% is distributed in coarse and bulky material.

Table 1. Characterization of bottom sediments

Grid	Sand particle diameter (mm)	Bulky particle diameter (mm)
D15	0.19	1.25
D50	0.28	1.65
D90	0.42	1.80

The seeding of sediment particles in the bottom of laboratory channel was established with an approximation by the PIC method (Particle in Cell), generalizing an initial concentration of sediments in a moment of time zero with 700,000 particles in this case. The concentration in an individual cell is obtained dividing the total mass in the cell by its volume.

The Fig. (8) presents the results of sediment transport for 1500 s in zero degrees of skewedness, in these images observed at different times (500, 1000 and 1500 s) the evolution of the transport in the bottom and consequently, the areas of sediment accumulation before and after the box culverts.



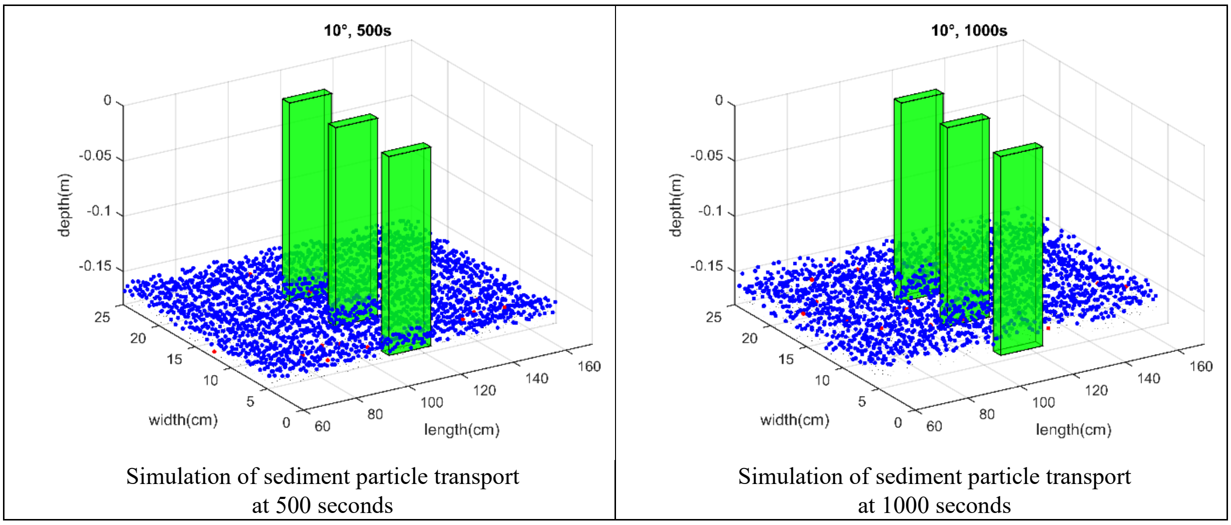
Simulation of sediment particle transport at 500 seconds

Simulation of sediment particle transport at 1000 seconds

Simulation of sediment particle transport at 1500 seconds

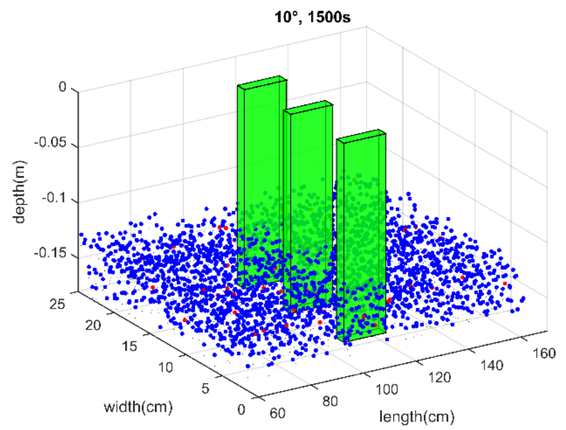
Fig. 8 Simulation of cloud evolution sediment particle transport to zero degrees of skewedness

The Fig. (9) presents the results of sediment transport in ten degrees of skewedness, we observed the different distribution by the hydrodynamic field near to bottom box culvert inlet, the areas of sediment accumulation were increased.



Simulation of sediment particle transport at 500 seconds

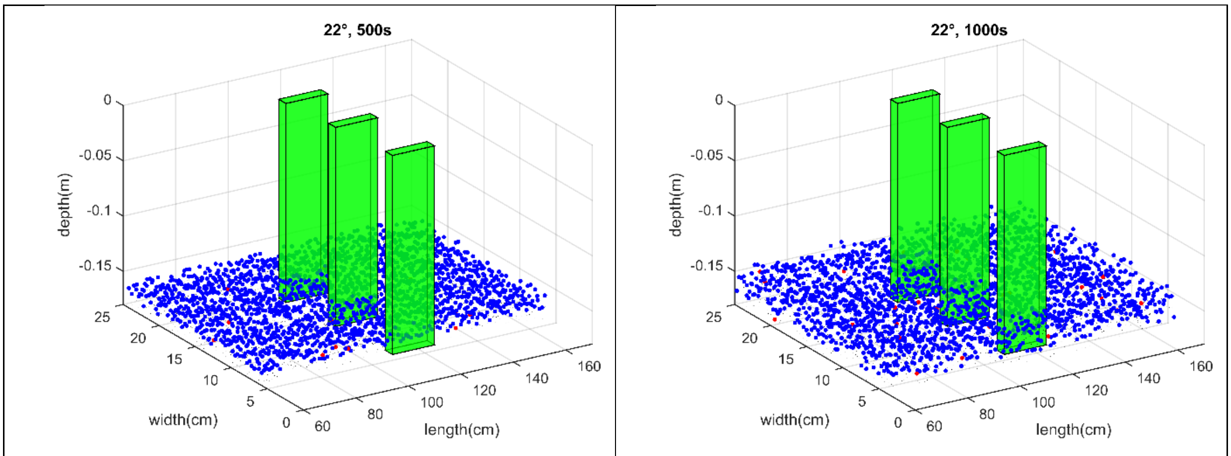
Simulation of sediment particle transport at 1000 seconds



Simulation of sediment particle transport
at 1500 seconds

Fig. 9 Simulation of cloud evolution sediment particle transport to ten degrees of skewedness

In twenty-two degrees of skewedness, the Fig. (10) presented low behaviour in distribution of the particle transport in the bottom, the areas of sediment accumulation were reduced by angle increment of box culvert position.



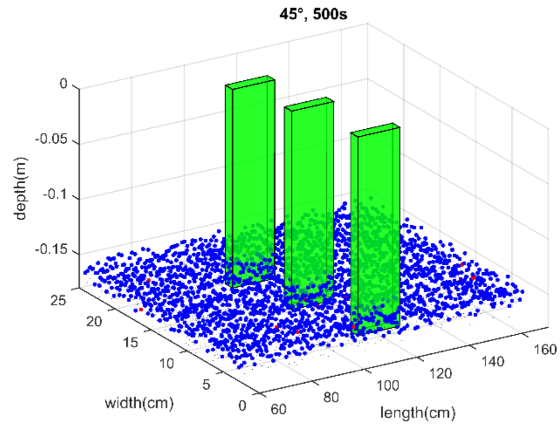
Simulation of sediment particle transport
at 500 seconds

Simulation of sediment particle transport
at 1000 seconds

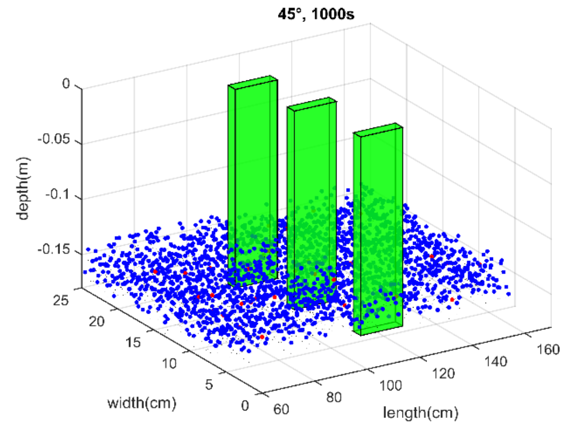
Simulation of sediment particle transport
at 1500 seconds

Fig. 10 Simulation of cloud evolution sediment particle transport to twenty-two degrees of skewedness

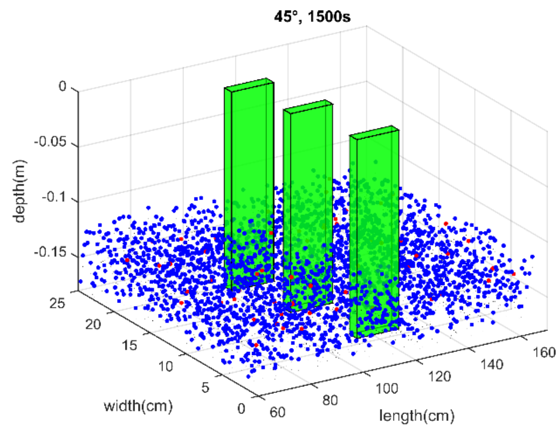
The Fig. (11) presents the results of sediment transport for 1500 s in forty-five degrees of skewedness, in these images observed at different times (500, 1000 and 1500 s) the evolution of the transport in the bottom and consequently, the areas of sediment accumulation before and after the box culverts.



Simulation of sediment particle transport at 500 seconds



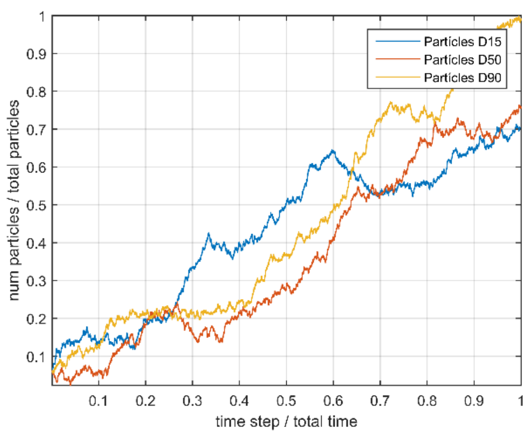
Simulation of sediment particle transport at 1000 seconds



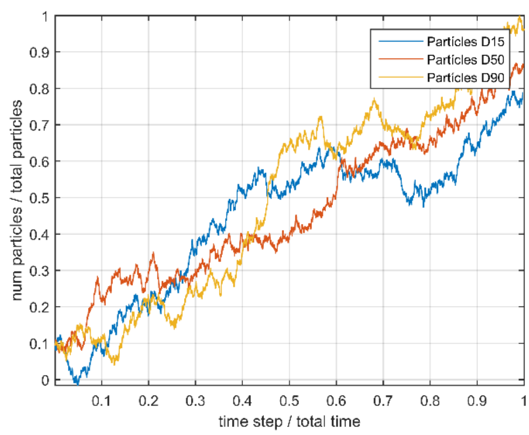
Simulation of sediment particle transport at 1500 seconds

Fig. 11 Simulation of cloud evolution sediment particle transport to forty-five degrees of skewedness

To visualize the behavior of the distribution of the number of particles of the three different diameters (D15, D50 and D90), in Fig. (12) the results of the distribution of the particles in the evolution of the simulation time are shown.



Results to zero degrees of skewedness



Results to ten degrees of skewedness

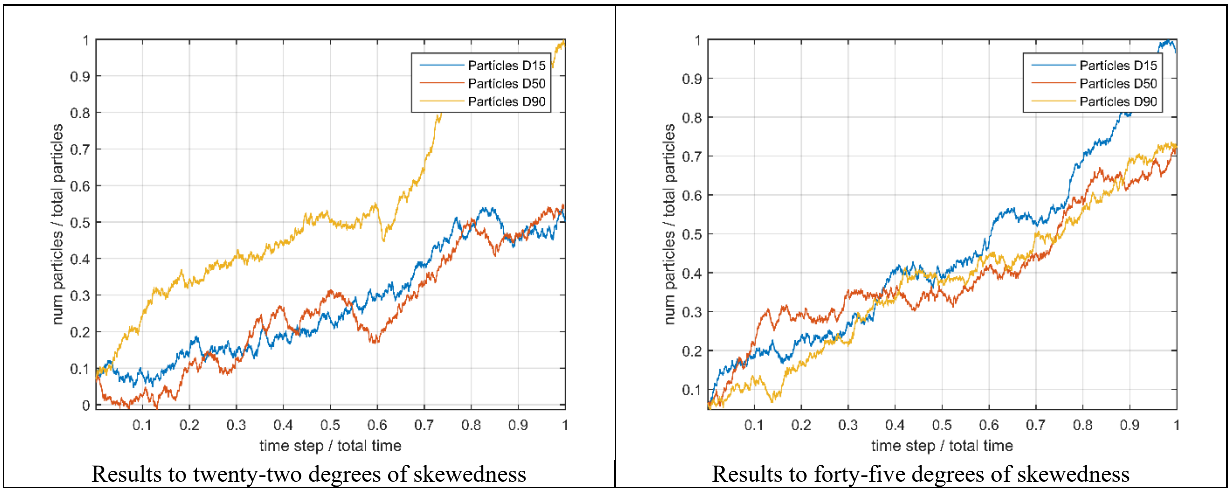


Fig. 12 Number of sediment particle diameters to different degrees of skewedness along to time simulation

The Fig. (13 to 16) shows the distribution of particles at the midpoint of sections A, B, C and D as a function of the number of particles in the water column represented as sediment concentration.

For the configuration of zero degrees of deflection, the results of the distribution of water column particles in the center of each section are shown in Fig. (13), the suspension of the background particles produced by the velocities of the flow in each section of the channel.

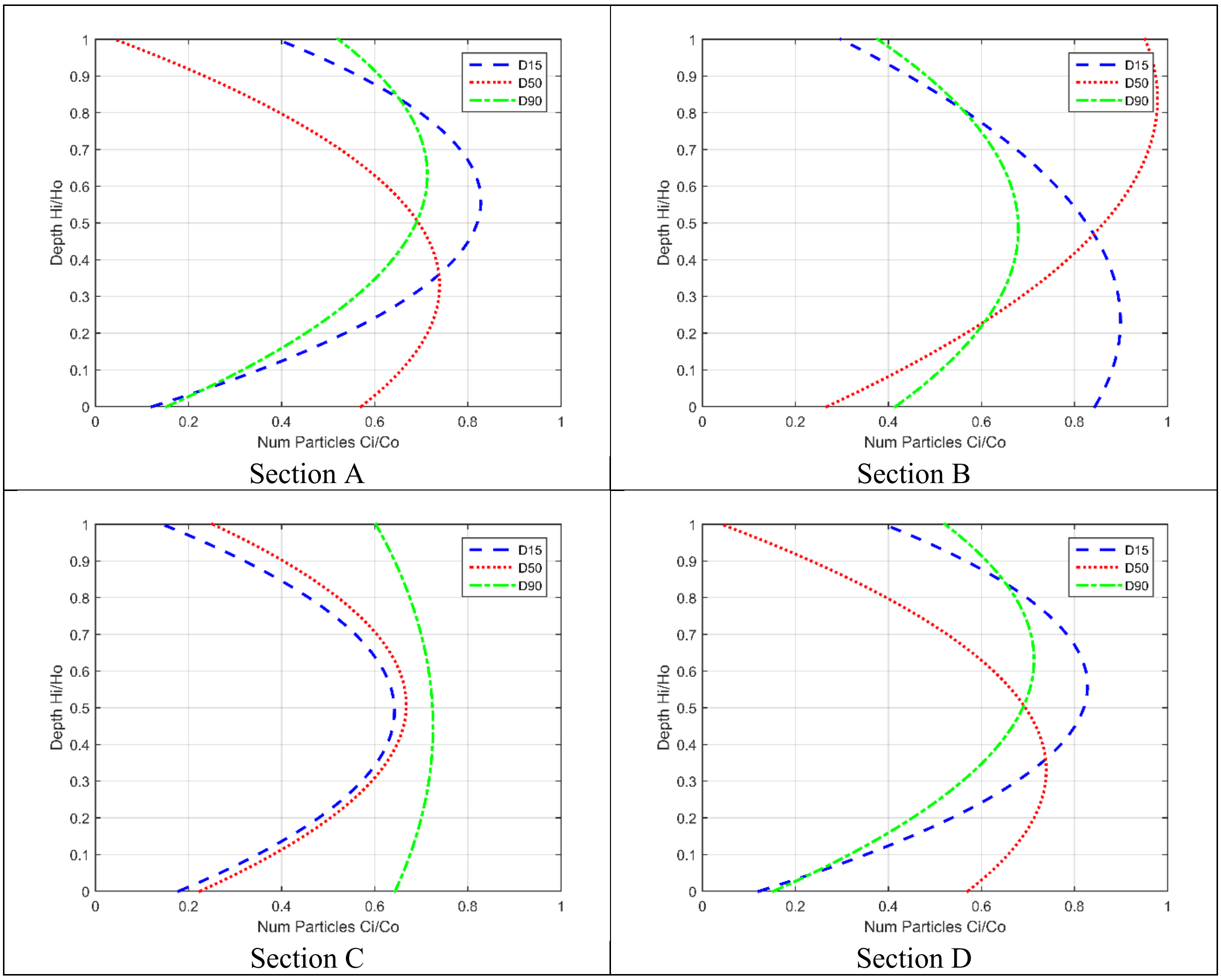


Fig. 13 Concentration profiles in all sections in zero degrees of skewedness at final time simulation

For ten degrees of skewed, in Fig. (14) the results of the concentrations of particles in each section are observed, sections B and C show values in D15 with representative changes in their concentration in water column, due to the randomness of the functions used and the speed field through these sections.

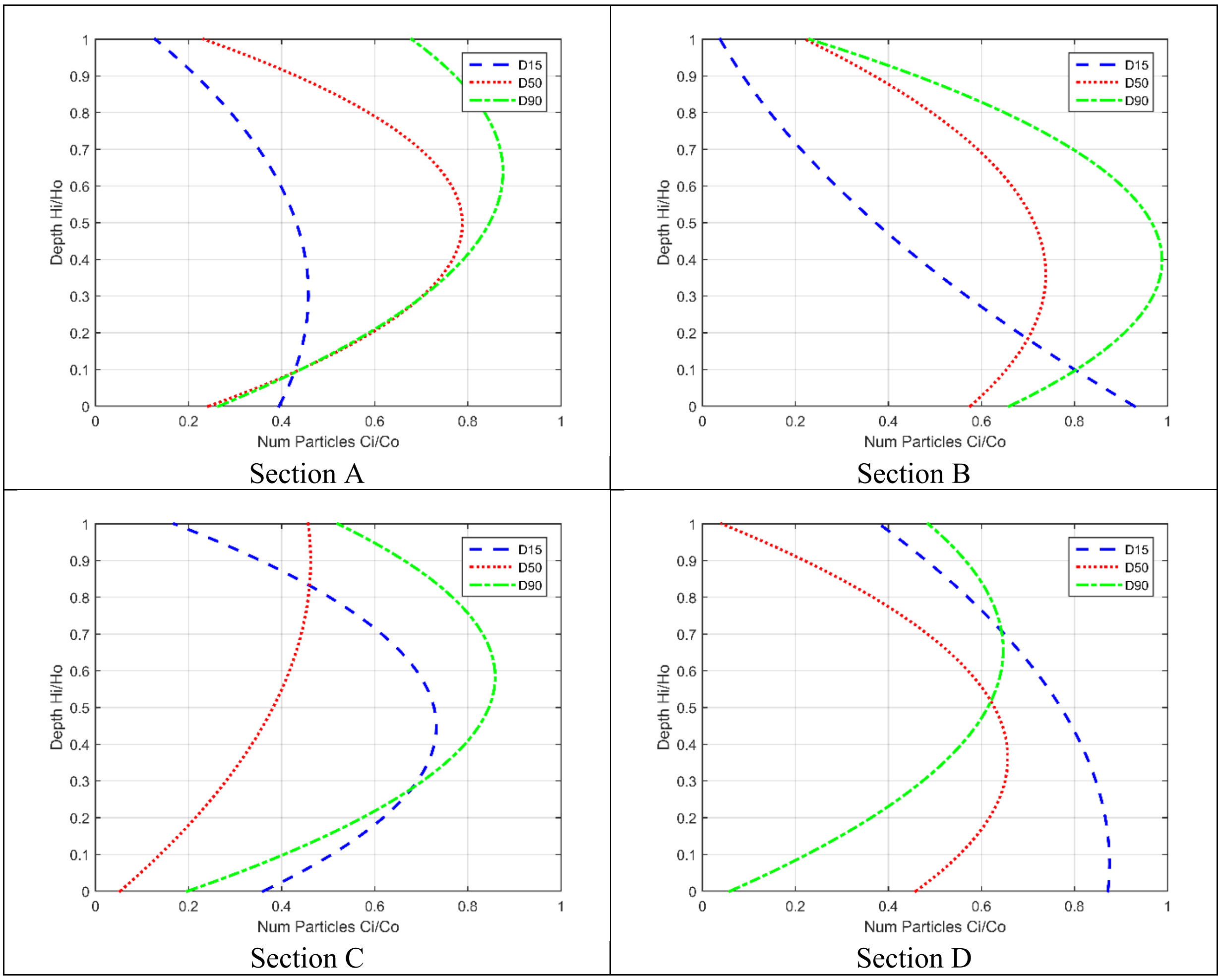
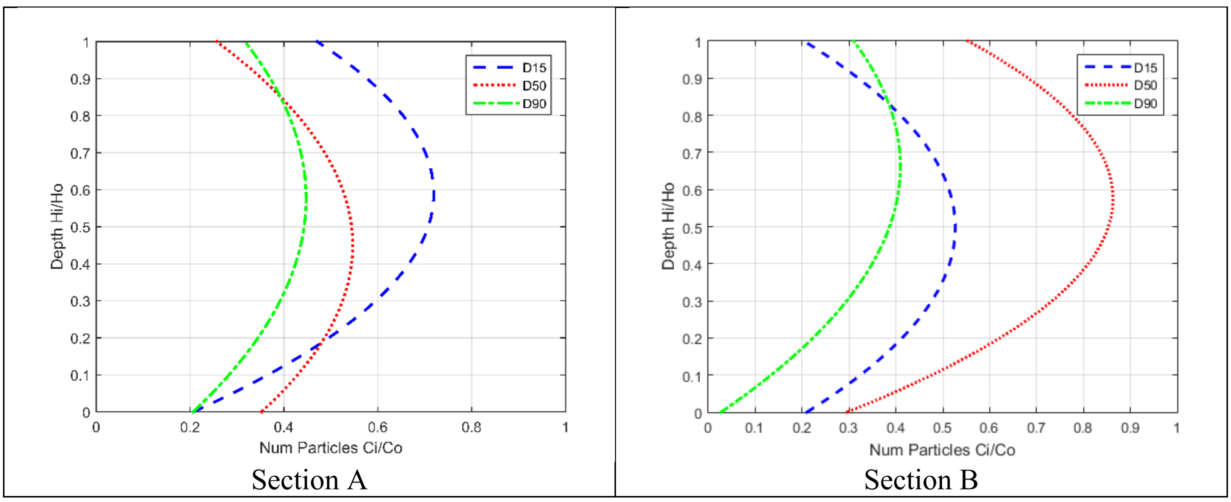


Fig. 14 Concentration profiles in all sections in ten degrees of skewedness at final time simulation

Similarly, in Fig. (15), we show the results for twenty-two degrees of skewed, where in section D, it presents a different behavior in the representative D15 in its concentration in water column, in relation to the other sections, It is estimated that it is due to the velocity field that passes through this section.



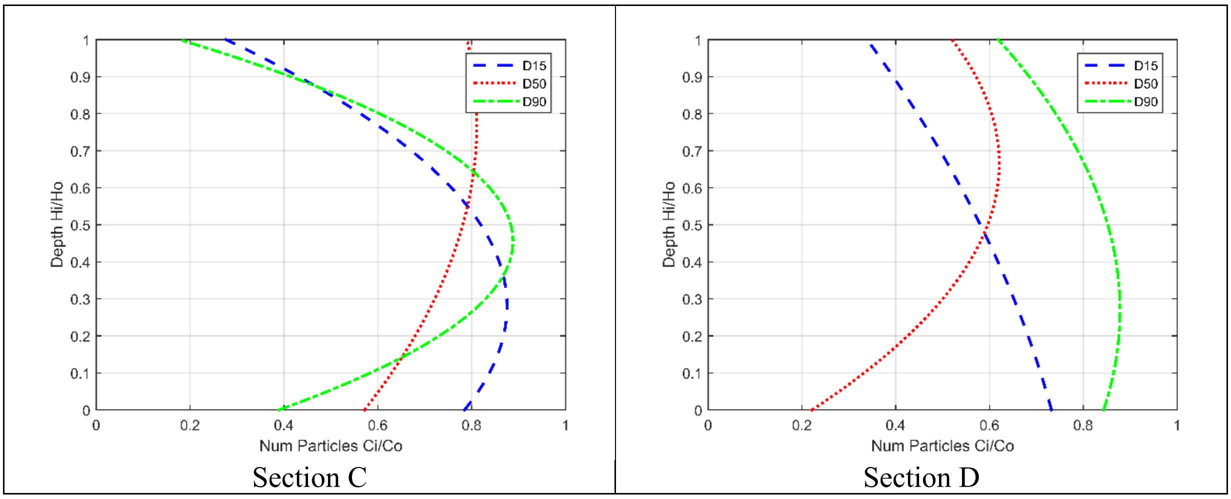


Fig. 15 Concentration profiles in all sections in twenty-two degrees of skewedness at final time simulation

Finally, in Fig. (16), the results for forty-five degrees of skewed are shown, where it is observed that the values of the concentration of particles in water column increase their value in suspension due to the velocity field that is presented in this configuration.

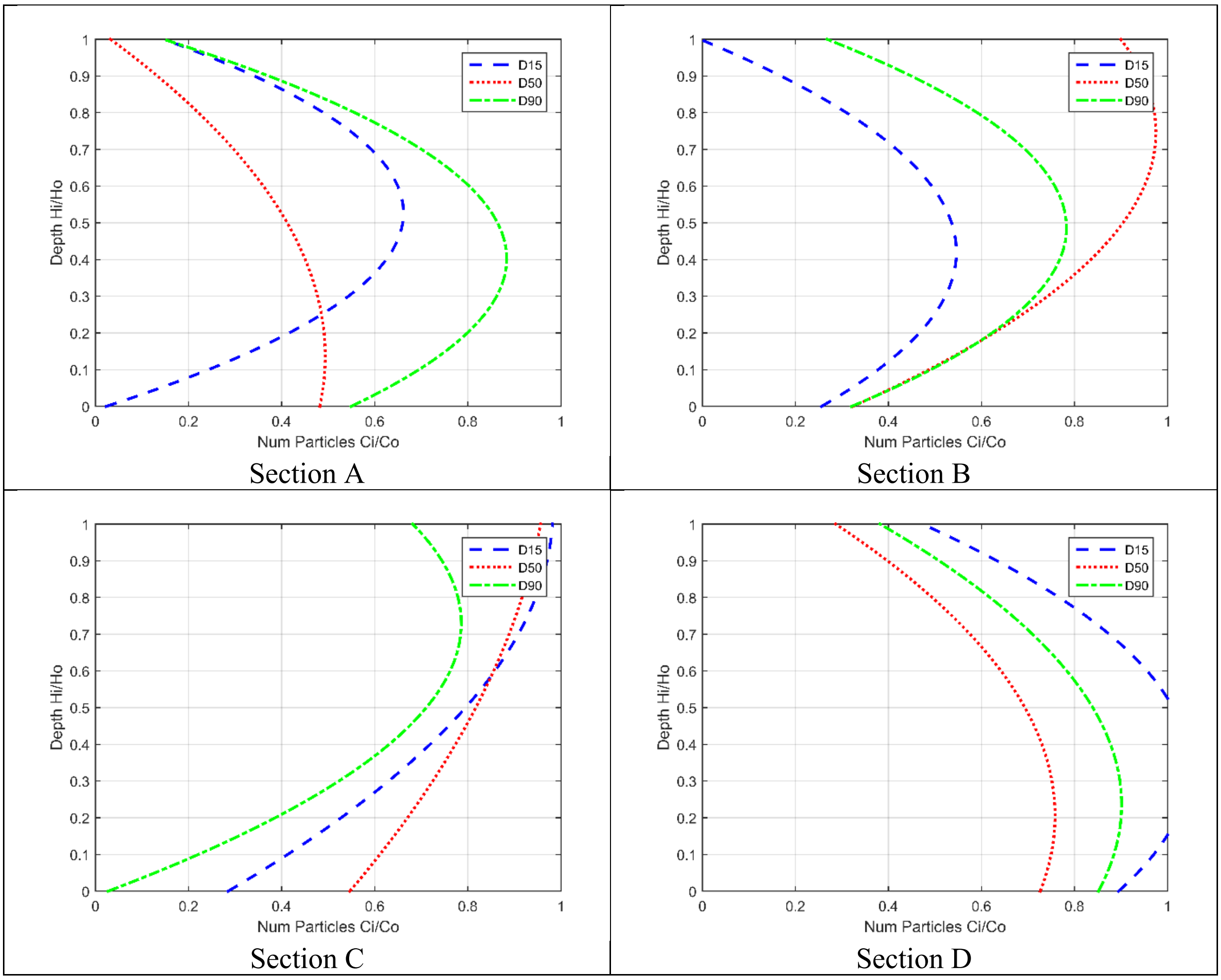


Fig. 16 Concentration profiles in all sections in forty-five degrees of skewedness at final time simulation

The results show that, for different degrees of skewed, they cause a different behavior of the particles in the water column, especially in those sections such as C and D which are further away from sections A and B which are the ones that they have a first contact with the flow.

CONCLUSIONS

A general model for the sediment transport and dispersion of pollutants has been developed, based on a Lagrangian formulation. The model allows to describe the evolution of tracer cloud both in the near and far-field.

The introduction of random motion to the equations of the particle movement produces good results by simplifying the equations of particle motion, thus producing a model that can withstand a large number of particles without consuming too many computational resources.

The model has the capacity to simulate 700,000 particles, limited only by the memory of the computer that is used. In this case, the evolution of tracer cloud is a function of the velocity field of the open channel and sediment process.

The results of the distribution of particles show considerable changes in the magnitude of concentration or number of particles in suspension that can be measured in a water column; due to the increase in the angle of skewed and the number of multi-barrel that are placed in the cross section, as well as the dimensions of each section and its geometry.

Actually, we are calibrating the numerical model with PIV measurements to 4 different scenarios of skewed multi barrel crossings.

Therefore, it may be concluded that sediment particle transport allows estimate which is the better configuration of skewed box-culvert to minimize suspension sediment particles and erosion in each cross section.

REFERENCES

[1] Rodríguez C., Couder C., Flores E., Herrera I.E., Cisneros R., "Modelling shallow water wakes using a hybrid turbulence model", Journal of Applied Mathematics, Vol. 2014, pp.1-10.

[2] Herrera I.E., Rodríguez C., Couder C., Gasca J.R., "Modelación numérica hidrodinámico-hidrológica en zonas de inundación con presencia de infraestructura", Tecnología y Ciencias del Agua, Vol. VI, Num. 1 Ene-Feb. 2015, pp. 139-152.

[3] Herrera I.E., Torres F.M., Moreno J.Y., Rodriguez C., Couder C., "Light particle tracking model for simulating bed sediment transport load in river areas", Mathematical Problems in Engineering, Hindawi Publishing, Vol. 2017, pp.1-15

[4] Gu H.B., Causon D.M., Mingham C.G. and Qian L., "A Fast-Marching Semi-Lagrangian level set method for free surface flows", Proc. of the Nineteenth International Offshore and Polar Enginnering Conference (ISOPE) 2009.

[5] R. J. Bialik, "Lagrangian modelling of saltating sediment transport: a review," in Rivers physical, Fluvial and Environmental Processes, GeoPlanet: Earth and Planetary Sciences, pp. 427–441, Springer International, Cham, Switzerland, 2015.

[6] S. Naqshband and B. McElroy, "Sediment transport at grain scale: a review, future research and morphological implications," in River Flow 2016, pp. 914–918, CRC Press, Boca Raton, Fla, USA, 2016.

[7] Broomans, P. "Numerical accuracy in solutions of the shallow-water equations", Master thesis, 2003, TU Delf & WL, Delf Hydraulics.

[8] Stansby, P. "A mixing-length model for shallow turbulent wakes", Journal of Fluid Mechanics, Vol. 495, 2003, pp. 369-384.

[9] Flather R.A., "Results from surge prediction model of the North-West European continental shelf for April, November and December", Institute of Oceanographic (UK), Report number 24, 1976.

[10] Robinson M., Monaghan J., and Mansour J. "SPH simulation of 2d wall-bounded turbulence", SPHERIC, Smoothed Particle Hydrodynamics European Research Interest Community, Second International Workshop 2007, 107–110pp.

[11] Hernandez I., "Modelos euleriano-lagrangeanos en flujos a superficie libre: Aplicación al transporte de partículas suspendidas y al crecimiento de microorganismos", Tesis Maestría, 2003, DEPFI, UNAM.

[12] Van-Rijn L.C., Walstra D.J. and Van-Ormondt M. "Unified view of sediment transport by currents and waves IV: Application of Morphodynamic model", Journal of Hydraulic Engineering 2007, 133-7:776– 793pp.