Journal of New Technology and Materials

JNTM

Vol. 7 N°01 (2017)53-63

OEB Univ. Publish. Co.

# The influence of obstacles and suspended matter on the distribution of turbulent flow velocities in a lamella separator

Farida Merrouchi[1], Ali Fourar[1], Fawaz Massouh[2], Djamel haddad[3] and Hanane Azoui[4]

[1] *University of Batna2, Hydraulics Department,Algéria*
[2] Laboratory Dyn-fluid E*NSAM, Paris- France*
[3] *University of Batna2,Mechanical Department,Algéria*
[4] PRIMALAB Laboratory, University of Batna 1, Algeria
Corresponding author: email: merrouchi.farida@yahoo.fr

Received date: September 22, 2016; revised date: May 20, 2017; accepted date: May 23, 2017

***Abstract***

*Lamellar settlers are used for stormwater treatment in order to separate suspended matter form water. In these settlers, the effective decantation area is increased thanks to the presence of many successive inclined plates.*

*The problem of lamella settling tank is the inhomogeneous distribution of flow velocity between the inclined plates so this work focuses on the study of turbulent flow and its interaction with the successive plates.*

*In this perspective, we propose a coupling of Euler and Lagrange approaches in order to model flow and sediment transport. The numerical model has been implemented in the Fortrancode. The finite volume scheme (ADI) was adopted as it ensures good precision with reasonable computation time. The model introduces the Lagrangian time constant CL to determine the size and lifetime of the eddies where the particles move.*

*For testing our program we first studied the case of turbulent flow around a single plate and then two plates where we found satisfying results.*

*Then we enlarged our study to study several plates. The obtained results show that the lamellar elements are efficient to reduce flow velocity and increase the sedimentation rate comparatively to conventional tanks with vertical obstacles or without barriers.*

*Keywords: obstacles, numerical modelling, turbulence, secondary currents, walls law, finites volumes, scheme ADI,lamella separator*

## 1. Introduction

The modeling of free surface flows are particularly situated at the level of the complexity of the met geometrical configurations, the variation of the shape of the bed, the inhomogeneous roughness of the bottom,hese flows are generally three-dimensional and unsteady.

The numerical difficulty increases with the presence of turbulence and the obstacles as for case example a lamellar decanter which is characterized by a significant number of obstacles

To understand the effect of one obstacle or several obstacles on the turbulent flow is a subject for a long time studied (e.g. Lamb (Lam1932)), Length (Lon1954). There is a particular large number of asymptotic analyses (Houghton and Kasahara [HK on 1968], Baines [Bai(Bay) on 1995], Dias and Vanden-Broeck [DV on 1989]) and digital (Lamb and Britter [LB on 1984], Lowery etLiapis [LL 1999]) who had for objective to determine asymptotic solutions of the shape of the free surfaces asymptotic solutions our study aims to be to know the local flow in the neighborhood of the obstacles in particular, when the free surface is far from these obstacles ( big heights) and the influence of the obstacles on the inhomogeneous distribution of velocity

Inclined parallel plates are a classical subject with a long history, and Boycott [1] was the first who observed that the settling rate of suspension is better “if the tube is inclined than when it is vertical

The settling behavior in inclined vessel was modeled firstly by Ponder [2] and later by Nakamura and Kuroda [3] and for this reason is known as the PNK theory.

In a primary sedimentation tank, where the discrete settling prevails, Imam et al [4] applied a fixed settling velocity and used an averaged particle velocity; there are very few theoretical studies about lamellar settlers. Doroodchi et al. [5] investigated the influence of inclined plates on the expansion behavior of solids suspensions in liquid fluidized beds

So far, many researchers have used CFD simulations to study water flow and solids removal in settling tanks for sewage water treatment

However, there are not many works in the literature in CFD modeling of sedimentation tanks for potable water treatment.[11]

In Euler-Lagrange models, the Navier-Stokes equations are solved for the fluid phase while Newton's equation of motion isused to model the particle phase motion. The trajectory ofeach particle is calculated from a balance of the forces acting on it(Maxey & Riley, 1983).

The continuous phase (water) are governed by the equations of Navier-Stokes and the equation of continuity .The numerical methods of resolution of these equations are limited until our days because by one hand there is no reliable and fast standard numerical method, and on the other hand, the resolution of the equations of Navier-Stokes governing the problem in general is very widely out of reach, where from the necessity of making approximations and of neglecting certain phenomena.

In this work for the continuous phase we suggest beginning a numerical simulation of the unsteady, turbulent and viscous flow of an incompressible fluid in the presence of obstacles in a channel to complex geometry

For flows that are not heavily loaded, such as those encountered in the case of decanter water in drinking water treatment plants, we neglect the interactions between the particles and the dynamic forces of the fluid govern the transport of the particles . The influence of the particulate phase on the fluid phase is also neglected because of the low particle concentrations (Dufresne, 2008; Yan, 2013) [6].

For the discrete phase We have taken only the solid particles which have a density greater than the density of the water and a shape of a sphere

The main aim of the present paper is to study the hydrodynamics and flow behavior of sedimentation with a system of inclined parallel plates (lamellar settlers) using the coupling between a Lagrangian method (discrete phase model) and Eulerien method Continuous phase.

## 2. Nature and position of the problem

In the present paper the sedimentation tank for potable water treatment plant whitch is constructed in the city of Timgad wilayaBatna

The treatment process is a gravity separation between two juxtaposed blades and inclined relative to a horizontal plane. In this type of separator, the raw effluent enters horizontally one of the lamellar block flanks. The water flows between the blades while The denser particles than the water descends . The clear waters come out the other side of the lamellar structure .

The velocity rates usually used to calculate the number of blades in the decanters are very low (Chebbo 1992[6] Chocat 1997[7] , Ashley et al. 2004[8] ). In practice, it means that it is necessary to implement very important surfaces of settling to be able to separate these very fine particles. Unfortunately, this does not facilitate the distribution of the water in all of the lamellar structure. But one of the assumptions used to calculate the number of blades is the homogeneous distribution of water among all lames.it is one of the keys to proper operation of a lamella separator This problem is also reported by Chocat (1997) [7] in his "Encyclopedia of urban hydrology and sanitation." the equal distribution of hydraulic flow problems on the slats are particularly difficult to solve and no really satisfactory solution on an industrial scale could be reached.[9]Then this article is to study the influence of obstacles on the repair of flow velocities in a lamella separator

## 3. Calculating assumptions

In this paper, we used the Euler–Lagrange multiphase approach with one-way coupled calculations where the movement of the particle is affected by the fluid phase but not the reverse.

In order to account for turbulent dispersion, the stochastic tracking of particles due to turbulence eddies was determined using the discrete “random walk model” in conjunction with RNG $k - \varepsilon$ model.Dufresneand al. (2009)[10] andRoza and al (2014)[11]

In this model, the lifetime and size of vortices Characteristics of the turbulence are calculated from a time constant Lagrangian CL. Dufresneand al. (2009)[10] showed that the default value (0.15) did not make it possible to reproduce the deposits observed experimentally.

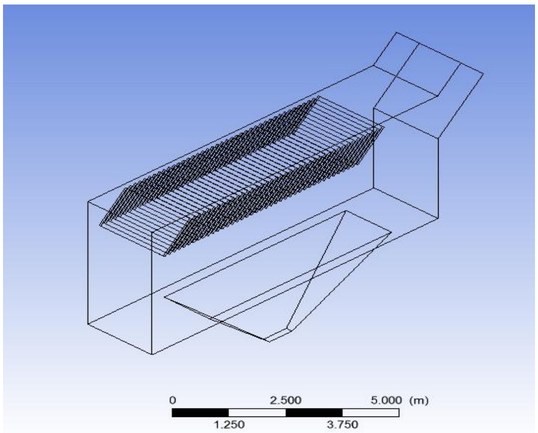


Figure 1: lamellar settler of water treatment plant

The decantation in the case studied is a decantation of the isolated particles but not flocculent particles (whose density and shape change during the decanting process), while the isolated particles are always present but with a low concentration [10],[11], For this reason and for the time which is very long with the coupling between the two approachesEulerian-Lagrangian especially for a 3D simulation, it has been assumed that the suspended materials are isolated particles which retain their density and which have a spherical shape.

The particle density used was 1080 kg/m 3 and the range of particle size was 30–890 μm and followed the Rosin–Rammler distribution [11].

## 4. Mathematical modelingof hydrodynamics and sediment transport in unsteady turbulent free-surface flow

The Euler-Lagrange approach for modeling of particle transport is used; this is for saying that the Navier-Stokes equations are solved for the fluid phase while the equations of Newton motion are solved for the particle phase in to determine particle trajectories.

### *4.1. Fluid phase*

Fluid phase is treated as a continuum by solving the Navier–Stokes equations so the equations of conservation of mass (Eq. (1)) and momentum (Eq. (2)) in the case of incompressible, instationary turbulence can be written in Cartesian-tensor notation as:

$$\frac{\partial \overline{u_i}}{\partial t}+\overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{P}}{\partial x_i}+v\frac{\partial^2 \overline{u_i}}{\partial x_k \partial x_k}-\left(\frac{\partial \overline{\grave{u}_i \grave{u}_j}}{\partial x_j}\right)+g(1)_i$$

$$\frac{\partial \overline{u_i}}{\partial x_i}=0 \qquad (2)$$

Where p is the static pressure, $v$ is the kinematic viscosity, ui denotes the instantaneous velocity associated with the x i coordinate direction, while U i is the average, mean flow velocity and u i′ is the turbulent velocity fluctuation such that u i = U i + u i

### *4.2. Turbulence*

The general transport equations for the turbulence kenetic energy $k$ and the turbulence dissipation rate Ɛ of the $k-\text{Ɛ}\ RNG$ [10], [11] turbulence model, can be described by equation (3) and equation (4) respectively

$$\frac{\partial k}{\partial t}+\overline{\mathrm{v}}_i.grad\ k = div.(grad\ k)+P-\varepsilon \qquad (3)$$

$$\frac{\partial \varepsilon}{\partial t}\overline{\mathrm{v}}.grad\ \varepsilon = div\left(\frac{v_t}{\sigma_\varepsilon}grad\ \varepsilon\right)+\frac{\varepsilon}{k}(c_{\varepsilon_1}p-c_{\varepsilon_2}p) \quad (4)$$

where k is turbulence kinetic energy due to mean velocity gradients, , μeff is the effective viscosity and $c_{\varepsilon_1}$, $c_{\varepsilon_2}$ are turbulence model constants.

The dispersion of particles due to turbulence in the fluid phase is predicted using the discrete random walk (DRW) model that includes the effect of instantaneous turbulent velocity fluctuations on the particle trajectories through the use of stochastic methods this approach, predicts the turbulent particles' dispersion by integrating the trajectory equations for individual particles, using instantaneous fluid velocity along the particle path during the integration. By computing the trajectory in this manner for a sufficient number of representative particles, the random effects of turbulence on the particle dispersion is accounted for.

### *4.3. Equation of particle motion*

The proposed model predicts the trajectory of a discrete phase particle by integrating the force balance on the particle, which is written in a Lagrangian reference frame. This force balance equates the particle inertia with the forces acting on the particle.

$$m_p\frac{d\overrightarrow{u_p}}{dt}=\overrightarrow{F_p}$$

$$\frac{dx_p}{dt}=u_p$$

Then our model of resolution is:

$\rho = c(x,y,z,t)\rho_{water} + \left(1-c(x,y,t)\right)\rho_{mes}$

$C(x,y,z,t) = 0$If the cell is filled the MES

$C(x,y,z,t) = 1$If the cell is filled with water

If $\boldsymbol{C(x,y,z,t) = 1}$

$\overline{NVS}+k+\varepsilon$

If $\boldsymbol{C(x,y,z,t) = 0}$

$\rho$=$\rho_{mes}$ if we solve the system below

Where: c: volume fraction, $\rho$:Density of the fluid [kg / m3], MES: suspended matter

The equations of Newton's movement are resolved for the particulate phase

$$m_p\frac{d\overrightarrow{u_p}}{dt}=\overrightarrow{F_p} \quad (5)$$

Knowing the sum of the forces acting on the particle $\overrightarrow{F_p}$the trajectory there of is calculated by integrating the velocity of the particle ,with the aim of determining the trajectories of particles. Trajectories are obtained by solving the following equation:

$$\frac{dx_p}{dt}=u_p \qquad (6)$$

The sum of forces acting on a particle moving in a viscous fluid is written

$$\overrightarrow{F_p}=m_p\frac{d\overrightarrow{u_p}}{dt}=\overrightarrow{F_D}+\overrightarrow{F_P}+\overrightarrow{F_g}+\overrightarrow{F_A} \qquad (7)$$

The drag force for spherical particles is calculated as follows:

$$\overrightarrow{F_D} = m_p \frac{18\mu}{\rho_p d_p^2} \frac{C_D Re_p}{24} (\vec{u} - \overrightarrow{u_p}) \quad (8)$$

Where $m_p$ is the mass of the particle, dp is the diameter of the particle, ρp is the density of the particle, ρ is the density of the fluid, μ is the dynamic viscosity of the fluid and ν the kinematic viscosity of the fluid. CD drag coefficient is obtained from the following equation

$$C_D = \begin{cases} \frac{24}{Re_p} \; si \; Re_p < 1 \\ \frac{24}{Re_p}\left(1 + 0{,}15 Re_p^{0{,}687}\right) si \; 1 \le Re_p \le 1000 \\ 0{,}44 \; si \; Re_p > 1000 \end{cases} \quad (9)$$

With $Re_p = \frac{d_p \|u-u_p\|}{\nu}$ (10)

For Re≥ 1000

$$\overrightarrow{F_D} = 3\frac{\rho}{\rho_p}(\vec{u} - \overrightarrow{u_p})2 \quad (11)$$

Gravity and buoyancy give: $\overrightarrow{F_g} = m_p \vec{g}\left(1 - \frac{\rho}{\rho_p}\right)$ (12)

The force due to the pressure gradient is expressed as follows: $\overrightarrow{F_P} = \frac{1}{6}\pi d_p^3 \vec{\nabla} P$ (13)

Finally the added mass force:

$$\overrightarrow{F_A} = \frac{1}{12}\pi d_p^3 \rho_p \frac{d\overrightarrow{u_p}}{dt} \quad (14)$$

In addition to all these forces, one also takes into account the influence of the turbulent nature of the flow on the particle. The dispersion of the particles due to the turbulent fluctuations in the flow is modeled using a dispersion stochastic (random walk). instantaneous speed of the fluid is defined as: $u_i = \bar{u}_i + u_i'$.

$\bar{u}_i$ is the average flow velocity and $u_i'$, is deducted from the local parameters of the turbulence, that is to say:

$$u_i' = \xi\sqrt{2k/3}$$

Or ξ is a random number according to a normal distribution and k is the turbulent intensity:

For 0≤ c≤1 we will solve the partial differential equation for the volume fraction

$$\frac{\partial c}{\partial t} + \vec{u}\nabla c = 0 \quad (15)$$

After one will overwrite the value of the volume fraction in the following equation

$$\rho = c_1 \rho_{water} + c_2 \rho_{MES} \quad (16)$$

Then we solve the system

$$\overline{NVS} + k + \varepsilon \quad (17)$$

So:

Continuity equation

$$\frac{\partial}{\partial t}(c_1\rho_{01}) + \frac{\partial}{\partial x_j}(c_1\rho_{01}U_j) = 0 \quad (18)$$

$$\frac{\partial}{\partial t}(c_2\rho_{02}) + \frac{\partial}{\partial x_j}(c_2\rho_{02}U_j) = 0 \quad (19)$$

Equation momentum

$\frac{\partial}{\partial t}[(c_1\rho_{01} + c2\rho_{02})U_j] +$

$\frac{\partial}{\partial x_j}[(c_1\rho_{01} + c_2\rho_{02}U_j)U_j U_i] = -\frac{\partial P^*}{\partial x_i} + [(c_1\rho_{01} +$

*c2ρ02)−ρ0∗gi+∂∂xjc1μ1+c2μ2+c1μt1+c2μt2∂Ui∂*

*xj+∂Uj∂xi−23∂∂xi[(c1ρ01+c2ρ02)k]*

(20)

Where $c_1$ :Volume fraction of water, $c_2$ : volume fraction of suspended matter (sand), $\rho_{01}$, $\rho_{02}$ are respectively the density of water and suspended matter.

Variants of ***k*– *ε*** model adapted to sub viscous layer (models RNG)Chatelier P [12] to model the turbulence in the lamella separator is proposed using the version called "low Reynolds number model $k - \varepsilon$ in which $c_u$, $c_{\varepsilon1}$ and $c_{\varepsilon2}$ are algebraic functions of the Reynolds number turbulent $R_{et}$ or near the wall in the computational domain toule.

## 5. Channel geometry and flow parameters

In this work we will study the turbulent free surface flows considering the presence of suspended matter and the influence of obstacles (one and two obstacles).This phase to test the program capacity.

Secondly, we will study the influence of several obstacles on the velocity of the flow in the channel complex geometry.

For the first case we consider a horizontal channel of 20 m long and 2 m height. Water flows from left to right at a Reynolds number of about $2.10^5$. The obstacle is located at 5 m from the channel entrance and its height is 1m. The turbulent boundary layer is fully developed.

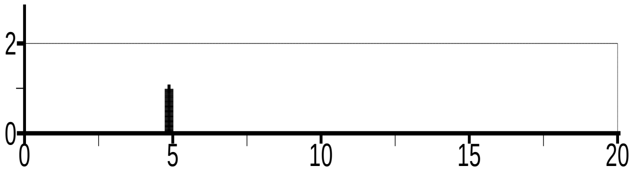


Figure2: channel with one obstacle

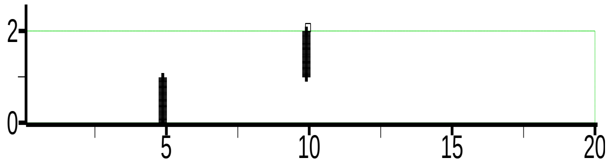


Figure3:channel with two obstacles

$$\underbrace{\frac{\partial(\rho\emptyset)}{\partial t}+\frac{\partial}{\partial x_j}(\rho\emptyset u_j)}_{1}=\underbrace{\frac{\partial}{\partial x_j}\left(\Gamma\frac{\partial\emptyset}{\partial x_j}\right)}_{2}+\underbrace{S_\emptyset}_{3} \qquad (21)$$

For the second case we consider lamella separator which is characterized by a wave height 1.16 m;Cutter width: 2m; spaces between blades:0.08m ;blade thickness: 0.07m ;Number of blades: 50 ;Material plates: PVC ,Blade tilt angle: 55 °
Dimensions of the settling tank: 9.8m long x 2m wide x 4m high.
Dimensions input channel: 2m long x 2m wide x 1m high

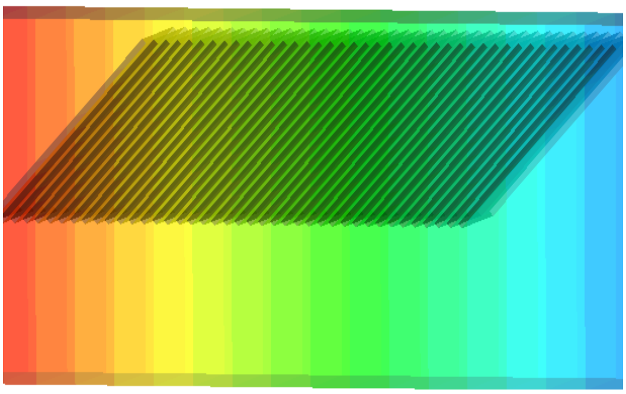

Figure4: The intermediate part (the lamellar part)

## 6. Numerical modeling with the method of finite volumes

All equations could be put in a general equation form of convection-diffusion for the variable Ø:

1-Convection term, 2- diffusion term, 3- source term

They will be resolved with initial and boundary conditions by finite volume method used by the code of the simulation.

*6.2.The solid phase*

Since the velocity of the fluid must be that at the position of the particle to calculate the drag force, a value $u_p$is interpolated from the values of the fluid velocity in the adjacent cells. The velocity of the particle is then calculated by discretizing the equation according to an implicit Euler scheme

$$\frac{u_p^n-u_p^0}{\Delta t}=-\frac{u_p^n-v_p^0}{\Delta t}+g\left(1-\frac{\rho}{\rho_p}\right) \qquad (22)$$

Where the exponent n refers to the values at time t + Δt and the exponent 0 to the values at the initial time

6.3. *Schematic implicit ADI*

The implicit methodis not restricted to the stability criterion, that is to say, it is stable for anyΔt ,and in other words it is unconditionally stable. One of these methods is the implicit method (ADI), it is very effective for solving the problems of transport phenomena.

6.7.*Adaptation of the time step*

The stability of the solution is ensured by adapting the time step. This oneIs adjusted at the beginning of the time iterationloop from the number of current)

$$C=\frac{u\,\Delta t}{\Delta x} \qquad (23)$$

Where c is the number of current ,Δt is the time step, **U**is the standard of the velocity through the cell and Δx is the size of the cell in the direction of the velocity .

6.*8. Boundary conditions*

For the discrete phase model the additional boundary conditions are: for the velocity inlet and pressure outlet an “escape” condition is prescribed.

Near the solid boundaries a “reflect” condition is prescribed

Finally, at the bottom of the tank a “trapped” condition is used.

For the continuous phase:

At the inlet of the decanter fixed value for velocity and ,turbulent kinetic energy,zero gradient for the pressure

Wall: fixed value for velocity=0 and ,turbulent kinetic energy,zero gradient for the pressure

Outlet:zero gradient for the velocity and turbulent kinetic energy ;zero gradient for the pressure

Finally we can conclude:

- A dynamic system sensitive.
- Use of ampirical laws.
- Boundary layer unmodelled.

## 7. Results

### *7.1.Case of one obstacle*

As considered, the presence an obstacle changes significantly the characteristics of the flow near the surface. Indeed, in the centre of the section, there is a relatively velocity associated with a very low turbulence intensity. On the other hand, at the obstacle, the flow properties differ.

It is very clear that the time required to see a fixed stationary regime is very long because the flow is totally renewed each time, this is due to the fact that the eddies created on the free surface and above the obstacle emerge Very quickly and slow down, c to say: small circulations.

An example of the result is shown in the figure (5) where it is seen that, beyond T = 40 s, the flow has been completely renewed without seeing the stationary steady state. The knowledge of typical time and of the mean value of the velocity U gives an idea of the size of the largest vortices contained in the flow.

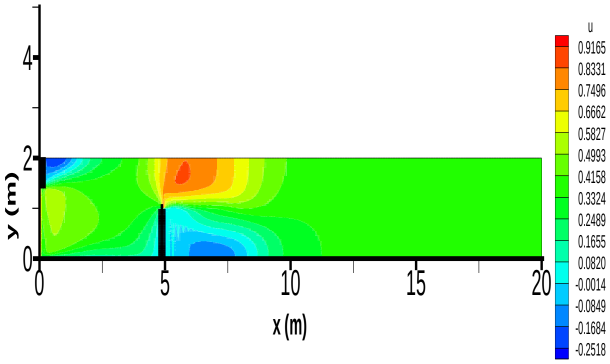


Figure5 - a: Temporal variation of the velocity field in m / s (T = 10 s)

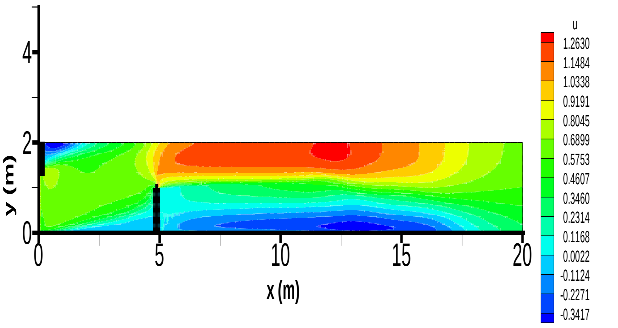


Figure5-b: Temporal variation of the velocity field in m / s (T = 25s)

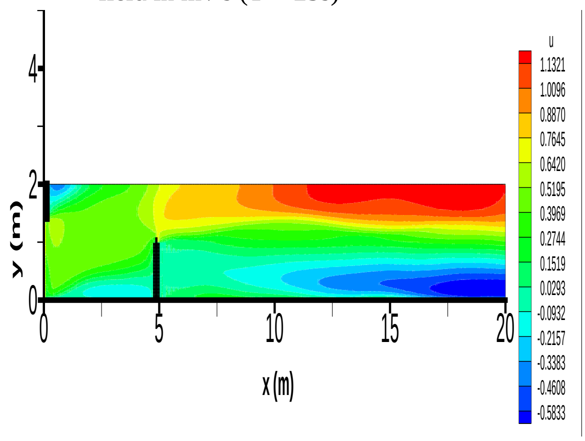


Figure5-C: Temporal variation of the velocity field in m / s (T = 35s)

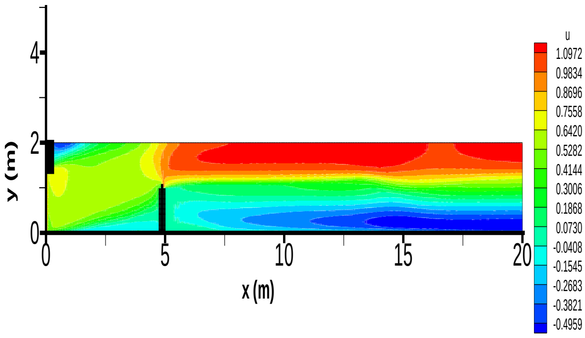


Figure5-d: Temporal variation of the velocity field in m / s (T = 40s)

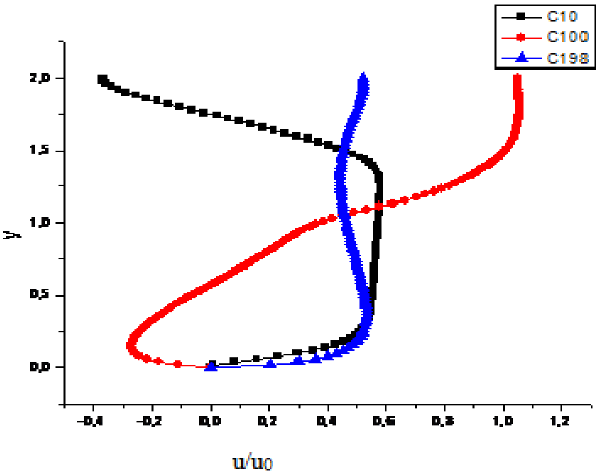


Figure 6: The velocity profiles in (m / s) calculated in a horizontal plane

In FIG. 6, the profiles of the speeds at the entrance before the obstacle c10 and after the obstacle c100 and far from the obstacle C198 can be seen, where the formation of the recirculation zones for the zone of l flow directly above the obstacle the penetration of the recirculation above the obstacle is respected, it can also be seen after the obstacle. The results obtained are in agreement with the specialized literature.

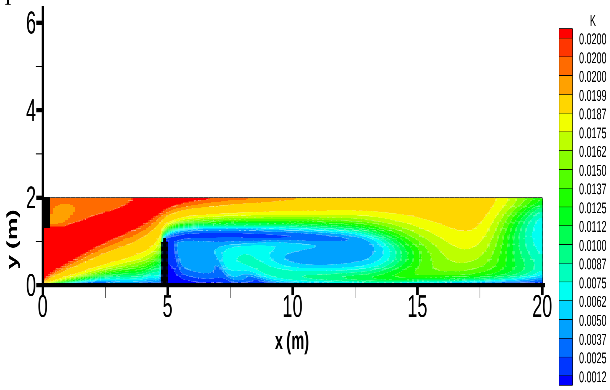


Figure7: Field of kinetic energy

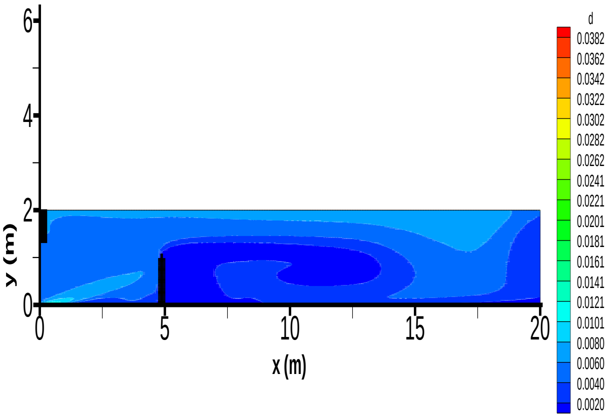


Figure 8: dissipation Field in ($m^2/s^3$)

The pressure profile in the presence of the obstacle is presented in the figure (9) where it can be seen that the pressure gradient is negative above the obstacle and downstream of the obstacle which corresponds to an acceleration Of the flow between the free surface and the obstacle created by the reduction in cross-section due to the presence of the obstacle .

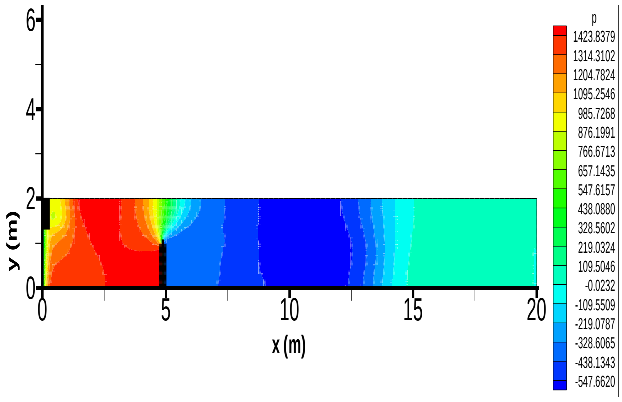


Figure9: Pressure field (pa) in a channel with one obstacle

For the area of the flow over the obstacle the intensity of the recirculation remains visible. With thek-εmodel we get an overestimation of the horizontal velocity component which leads to an increase of the turbulent kinetic energy.
This same type of result where we can see an increase in the turbulent mixing in the shear layers thus an increasing of the wake training rate was obtained by (Dragent, 1996) [13]
Despite the too diffuse nature of the model which is always underestimated the length of recirculation, but good results have been obtained (downstream of the obstacle), as the boundary conditions at the walls have been improved.

*7.2.Case of two obstacles.*

Interacting flows and turbulent flow processes, which are produced by the presence of the obstacle, cause energy losses;they are distributed downstream of the obstacle, and take place in shear layers, vortices, separation zones and turbulent decay in the wake [14]
The pressure profile in the presence of two obstacles is shown in Figure (10-a). We can say that we have a positive pressure gradient just the lover of the obstacle 1.
The positive gradient just greater prefers the barrier 2 is therefore two zones of overpressure (over short distances) and a slight negative pressure gradient in the area that lies between the two barriers and after the obstacle 2. Reflecting the slowdown front of the obstacle to the point

of stopping lover, which is consistent with the relationship Bernoulli, although it neglects viscous effects.

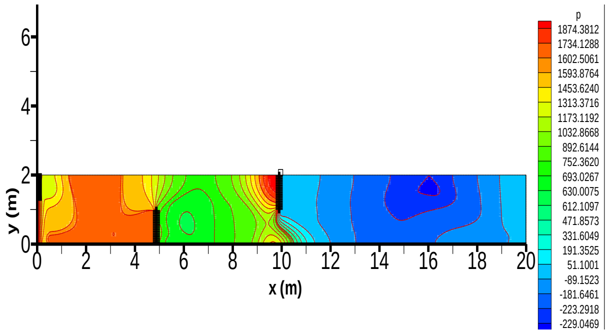


Figure 10-a: Pressure field (Pascal)

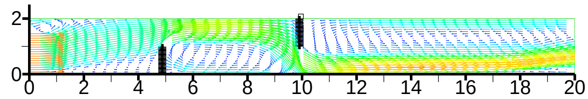

Figure 10-b:velocity vectors (m/s)

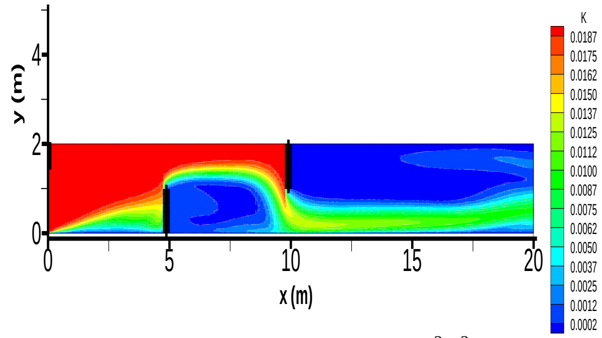


Figure 10-c: Field of kinetic energy in ($m^2/s^2$

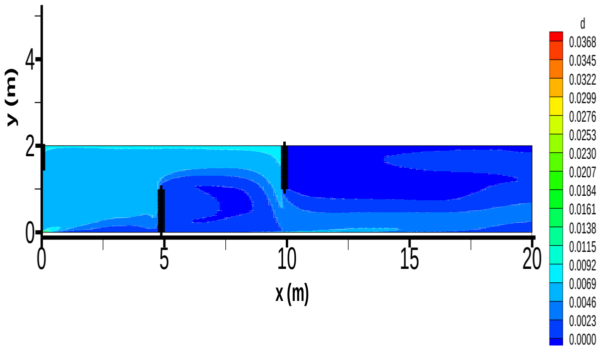


Figure 10-d:Dissipation Field in (m3/s3)

The presence of obstacles allows to obtain speeds of less important traffic. But with the model was an estimate of the horizontal component of the speed which leads to an increase of the turbulent kinetic energy and a better result for the length of recirculation.

*7.3Case of lamella separator(case of suspended matter)*

*7.3. a.Hydrodynamic fields in a lamellar settling*

*7.3. a.1Macroscopic observations*

The most important macroscopic observation drawn from this study is the influence of lamellar settlers on the flow field.;Examination of the simulations in a longitudinal and vertical median plane (Figures N. 11 and 12 and 13)

In the figure 11 At the initial injection (at t = 1 s), we observe an insignificant cloud of pollutant and the concentration increases in the vicinity of the entrance and the pollutant dispersed in both directions (horizontal and vertical) of the flow

At t = 2 s and t = 3 s, we can see that the position of the maximum concentration evolves in the longitudinal direction near the entrance and the pollutant continues its dispersion along the channel,which is in agreement with the specialized literature [14]

For the other figures : simulations show that it forms an important swirl in the vertical plane with lowering of the water lines to the back of channel, then lift water into the blades. They also show that water penetrates to the bottom

of the book, struck the rear compartment layered veil, then back in the opposite before recovering in the blades.

The formation of turbulence by the blades depends on the speed of water penetration. The main turbulence is approximately in the first half of the compartment. More work is, the more the turbulence lengthens. This turbulence seems to be causing the return movement of water in the opposite direction in the first blades.

If we observe the movement of water over the blade, the higher the compartment is, the more the area where the water flows against the direction is important. At the sludge storage area,slowly towards the bottom of the device without creating strong disturbances..

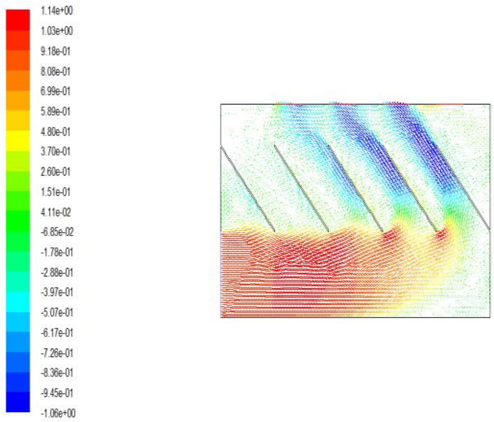

Figure11: velocity fields the particular phase in the first (five strips)

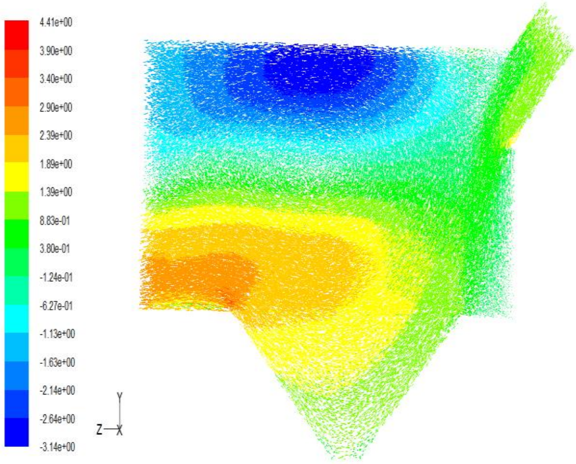


**Figure12:** velocity fields in the decanter along a longitudinal vertical plane YZ

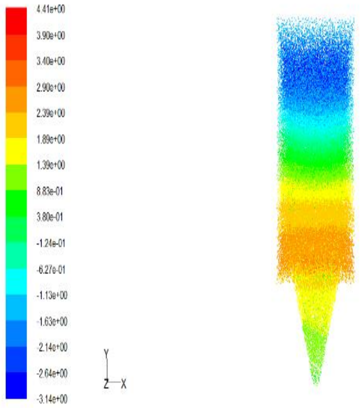


**Figure 13** : velocity fields in the decanter in a vertical plane

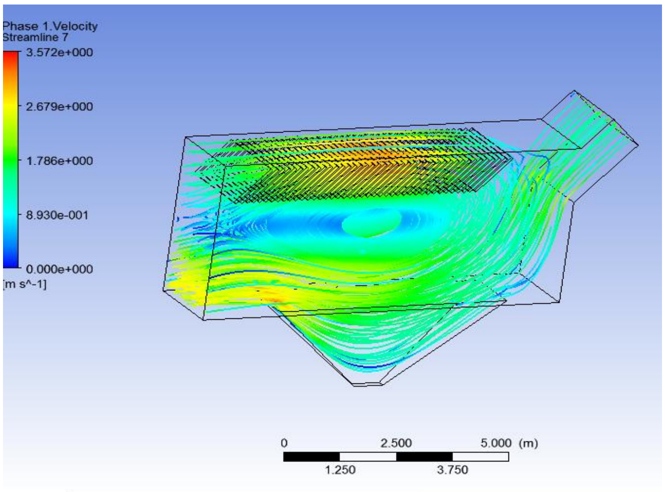


Figure 14: velocity streamline phase 1

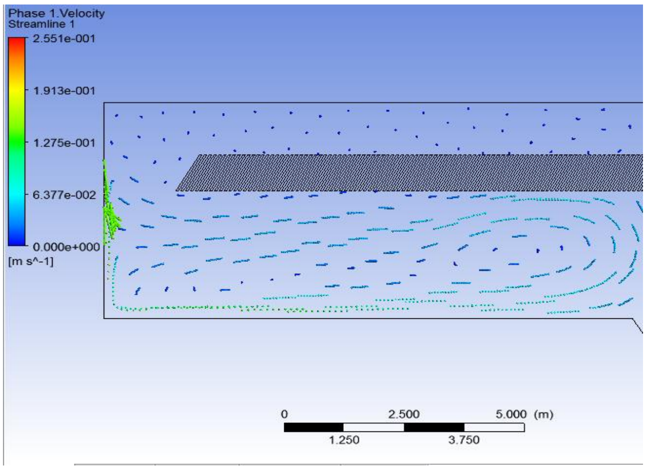


Figure 15: velocity streamline phase one and phase 2 (mes)

In Figures N. 14, 15 a representation of the flow in the three-dimensional blades. This figure confirms that the velocity distribution is not uniform throughout the lamellar structure. The flow rates are nevertheless low (a few centimeters / second). The closer the downstream threshold, the higher the flow velocity between the blades increases. The highest values of the speed (in red) are observed along the axis of the channel, and recirculation zones are observed singularities of the channel (the change of the section). The lowest values of the speed (blue) are noted along the walls, the blue area indicates a fluid recirculation following the roughness of the walls, the abrupt changes in section and the channel slope the raw effluent enters the feed channel with a speed of about 0.5 m / s. The flow strikes the veil placed down the hall, and then reverses direction, creating a large recirculation in the inlet channel. In this configuration, it will be difficult to have a uniform flow distribution in each group of slides

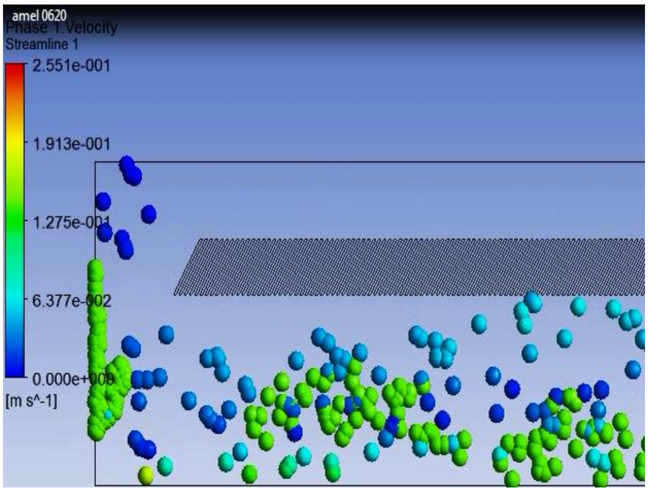


Figure 16: Velocity of deposit for different particle sizes

This figure (16) shows that a group of these particles is suspended in the flow (particles with a diameter 200 µm with a very low deposition rate while another group of these sediments is deposited Below the obstacles (particles with a diameter380 μm . However, particles of very small size remain suspended in the flow and get to the exit of the sewer without any deposition rate. Thus, we conclude that by increasing the particle sizes, the deposition rates increase.

In this study, we conducted a simulation on a model complicated by its geometry and boundary conditions for turbulent multiphase flow model. a lamella separator treats 1.57 m3 / s, the lamellar structure is calculated on the basis of a drop in speed of 1 m / h and a height under blade is 1.70 m,

The results of this simulation are satisfactory, they describe the flow taking into account the elongation of the settler compartment, the dimensions of the channel (port) power remain constant, these results in an increase of the flow velocity under the blades. If the velocity of the water under the blades is high, for example greater than 20 cm / s, it generates whirlpools not conducive to the formation of a sludge blanket on the raft.

There is a part of the flow recirculation contraflow in the blades placed above the vortex, and the flow rates through the blades are not homogeneous. The flow is faster gradually as it approaches the outlet. At the level of the floor, in a horizontal plane water descends gradually to the bottom of the decanter. Arrival against the veil downstream compartment, the water returns to misinterpretation and then up in the blades. The flow rates are however low, of the order of 5 cm / s which reveals the instability of water bodies.

## 8. Conclusion

In many industrial situations in nature, it is necessary to transport mixtures of different fluids or fluids and solids under extremely varying proportions,

This study focused rightly on the coupling between turbulence and obstacles (blades) and suspended matter

The Euler-Lagrange approach is chosen to simulate the flow and the sediment transport .improved by new conditions taking the particle properties ( différent diameter) .

for the turbulence , the decomposition of Reynolds equations makes the open system of equations and therefore to perform its closure it has been necessary to adopt a turbulence model able to handle turbulent flow in a lamella separator. It is interested focusing the attention on the models k-ε RNG. The RNG k-ε model seems the most appropriated for the flows with low Reynolds number (Hunze, 2008; Yan, 2013[4], [15], [16] ,[17]. This model takes account of the transport of turbulent quantities in their associating differential equations of transport

The three-dimensional modeling of the decanting structures allows us to understand the sedimentation process

It is common that part of the stream recirculated to misinterpretation in the blades placed on top of the vortex, and that the flow rates through the blades are not homogeneous.

The flow is faster gradually as it approaches the outlet. At the level of the floor, in a horizontal plane , water descends gradually to the bottom of the decanter. Arrival against the veil downstream compartment, the water returns to misinterpretation and then up in the blades. The flow rates are low, however, indicating the instability of the water bodies.

The flow is faster gradually as it approaches the outlet., which is in agreement with the specialized literature

The effective surface area for settlement is increased by the inclined plates giving a smaller footprint than in the conventional tank, and the installation of the lamellar settlers make the rectangular tank more efficient and more cost-effectiv

Finally, numerical simulations channels with barriers were used to test the concept of lamella separator. Numerical models manage to predict whether a work functions satisfactorily hydraulic viewpoint.

## References


[1] A.E. Boycott, Sedimentation of blood corpuscles, Nature (.a guaranteed hydraulic operation and therefore treatment, prior to the execution of a work(1920) 104–532

[2] E. Ponder, On sedimentation and rouleaux formation, Q. J. Exp. Physiol. 15 (1925)235–252.

[3]N. Nakamura, K. Kuroda, La cause de l'acceleration de la vitesse de sedimentation desuspensions dans les recipients inclines, Keijo J. Med. 8 (1937) 256–296.

[4] E. Imam, J.A. McCorquodale, J.K. Bewtra, Numerical modeling of sedimentationtanks, J. Hydraul. Eng. ASCE 109 (1983) 1740–1754.

[5] E. Doroodchi, K.P. Galvin, D.F. Fletcher, The influence of inclined plates on expansion behaviour of solid suspensions in a liquid fluidised bed— acomputationalfluiddynamics study, Powder Technol. 156 (2005) 1–7.

[6] Chebbo G. Solides des rejets pluviaux urbains. Caractérisation et traitabilité. Thèse de doctorat, Ecole Nationale des Ponts et Chaussées,(1992). 413 p

[7] Chocat, B. (1997). Encyclopédie de l'hydrologie urbaine et de l'assainissement, Bassins de retenue p. 95, Eurydice 92, Ed Tec&Doc Lavoisier, Paris, 1997, 1121 p.

[8] Ashley R.M., Bertrand-Krajewski J.-L., Hvitved-Jacobsen T., Verbanck M., (editors) (Solids in Sewers. London (UK): IWA Publishing, (2004).

[9] José Vazquez*, Antoine Morin**, MatthieuDufresne*, JonathanWertel*A CFD approach for shape optimization of lamellarSettlers NOVATECH 2010

[10] Dufresne, M., Vazquez, J., Terfous, A., Ghenaim, A. & Poulet, J.B. . Experimental investigation and CFD

modelling of flow, sedimentation, and solids separation in a combined sewer detention tank. Computers & Fluids (2009)38(5), 042-1049.

[11] RozaTarpagkou.The influence of lamellar settler in sedimentation tanks for potable watertreatment— A computational fluid dynamic study Powder Technology 268 (2014) 139–149

[12] Chatelier P. « Simulation de l'hydrodynamique des chenaux d'oxydation par l'utilisation des équations de Navier -Stokes associées au modèle $k - \varepsilon$ : évaluation de Technical Report n° 14, May 2004, 360 p. ISBN 1900222914.

[13]Dragent C. « Contribution à la modélisation de la dispersion de polluants : Etude de sillages autour d'obstacles de forme parallélépipédique ». Thèse de INP de Toulouse.5(1996).

[14] J.D. Fenton , Obstacles in streams and their roles as hydraulic structures, in: Proceedings of the 2nd [6] [6] International Junior Researcher and Engineer Workshop on Hydraulic Structures, Pisa, Italy, 2008, pp. 15–22 .

[15] Hunze M. (2008). Investigation of clarification tanks using 3-dimensional simulation studies- part 1 circular clarification tanks. Hannover, Germany, Flow Concept GmbH

[16] Yan, H., LipemeKouyi, G., Bertrand-Kraj ewski, J-L. (2011). Modélisation numérique 3D des écoulements turbulents à surface libre chargés en polluants particulaires dans un bassin de retenue-décantation des eaux pluviales. La Houille Blanche n°5, 40-44.

[17] Yan H. (2013). Expérimentations et modélisations tridimensionnelles de l'hydrodynamique, du transport particulaire, de la décantation et de la remise en suspension en régime transitoire dans un bassin de retenue d'eaux pluviales urbaines. Thèse de doctorat en hydrologie urbaine. Lyon. INSA de Lyon, 238 p.