Flow hydraulic simulation through two sand traps, using Ansys fluent

C D Rodríguez¹, and J S De Plaza¹
¹ Facultad del Medio Ambiente y Recursos Naturales, Universidad Distrital Francisco José de Caldas, Bogotá, Colombia

E-mail: cdrodriguezb@correo.udistrital.edu.co, sjsebastiand@correo.udistrital.edu.co

Abstract. Computational fluid dynamics is a tool that allows to simulate and observe the behavior of any fluid, based on a physical, hydraulic, and hydrodynamic analysis. This research analyses the behavior of the flow in a sand trap, which is a structure used to remove sand particles with a minimum size of 0.10 mm, prior to treatment in a drinking-water plant. The objective of this study is to determine the highest efficiency between two sand traps, one with a double smooth screen and the other with a double perforated screen (with diffusers), based on the simulation and analysis behavior of the flow inside each sand trap. The methodology used includes the traditional design of each unit based on Hazen's model and Stokes viscosity law, to later carry out the numerical model simulation from Ansys Fluent (pre-processing, processing, and post-processing). The result shows that perforated double screen sand trap generates a removal efficiency of 78%, while the smooth double screen 28%. In addition, other four units of interleaved screens are proposed, in these cases efficiencies of up to 50% are observed and it is shown that it is necessary to implement at least two perforated screens (with diffusers) to guarantee an efficiency greater than 70%. Hydraulic simulation has a broad impact on infrastructure works and consulting.

1. Introduction
One of the fundamental structures implemented in drinking water pre-treatment, is the sand trap (ST), this structure allows to carry out the solids’ removal of a certain diameter, as required, in order to facilitate the treatment in the plant [1]. This hydraulic type of structure can be simulated using computational fluid dynamics (CFD) [2], through this tool it is possible to obtain an estimated visualization of the flow behavior inside the ST and hydraulic operation analysis of the ST [3]. ANSYS is a software specialized in simulation, and within its components it is possible to perform a high precision CFD type simulation [4].

In this way, it is proposed to carry out the evaluation with two different types of ST, one with a double perforated screen and the other with a double smooth screen, and thus determine the maximum efficient in removing sand [5].

The determination of the best design depends on the analysis of the physical variables and the flow behavior inside the ST, however, the determining factor is the one that measures the removal of solids [6], since this is the one that defines the ST efficiency. Likewise, it is essential to consider all the physical phenomena present, such as velocity flow and turbulence to implement a structure with high removal efficiency [3,6]. Determining which unit, ST, is more efficient, prior to its construction, allows optimizing costs and foreseeing improvements for future problems.
2. Methodology and materials

The applied methodology consists in design development, then elaborate the 3D structures modeling and later, moving on to the development stages in the software, which are pre-processing, processing and post-processing, and finally get the data and analyze it.

2.1. Sizing and 3D model

The ST sizing is made from Stokes Law and the application of Allen Hazen’s model, as appropriate [7]. These models are based on the particle`s sedimentation rate and the unit (ST) efficiency [8,9]. See the Equation (1) and Equation (2).

\[ V_s = \frac{\rho_s - \rho_w}{18 \cdot g} \cdot d_s^2. \]  
\[ \%Ef. = -0.0374Hz^6 + 0.7669Hz^5 - 6.2939Hz^4 + 26.858Hz^3 - 66.116Hz^2 + 101.76Hz, \]  

where \( V_s \) is sedimentation velocity, \( g \) is gravity acceleration, \( \rho_s \) is particle density, \( \rho_w \) is fluid density and \( d_s \) is particle diameter. With the dimensions obtained and considering what is stipulated mainly for the analysis, the design is carried out in three dimensions using AutoCAD software, applying the tools indicated for 3D modeling. Once the model is elaborate in AutoCAD, it is exported to a sat format, since in ANSYS this format turns out to be compatible to perform the analysis.

2.2. Pre-processing

In this section, the analysis volume is defined from the generated geometry, from which the model in study is meshed. Then the relevant mathematical model is produced, which in turn is composed of the turbulence, discrete phase, and flow motion models.

2.2.1. Defining the analysis volume from the geometry and meshing. The analysis volume is defined from the geometry, flow limits, and the faces that function as contact surfaces. Subsequently, the meshing is done defining a cell size of 10 mm and tetrahedral type for greater precision, this also confers a more stable model from the courant number [9].

2.2.2. Turbulence model. The K-epsilon model is implemented like a turbulence model; this model represents the flow turbulence and turbulence dissipation properties through Equation (3) and Equation (4) [10], which are defined as follows.

\[ \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + \rho_e - \gamma_m + S_k. \]  
\[ \frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_\epsilon \frac{\epsilon^2}{k} + G_\epsilon + C_3 \gamma G_b - C_2 \rho \frac{\epsilon^2}{k} + S_\epsilon, \]

where, \( G_k \) is Generation of turbulent kinetic energy due to mean velocity gradients, \( G_b \) is generation of kinetic energy due to buoyancy, \( \gamma_m \) is contribution of fluctuating dilation in compressible turbulence, \( C_{1\epsilon}, C_{2\epsilon}, C_{3\epsilon} \) are experimentally determined constants, \( \sigma_t \) is turbulent viscosity, \( \sigma_k \) is Prandtl number as a function of \( k \) and \( \sigma_\epsilon \) is Prandtl number as a function of \( \epsilon \). First, the energy variable conditions and determines the turbulence variable, like the K-epsilon model [9,11].

2.2.3. Discrete phase model. Based on this model, the data necessary to simulate the particles are established, contemplating the fluid temperature, sedimentation velocity and its diameter [11], therefore,
according to what was proposed by Newton, the rotational and translational particle movements are described in Equation (5) and Equation (6).

\[ \frac{m}{dt} \frac{dv}{dt} = mg + \sum F_c + F_d + F_b, \]  
(5)

\[ I \frac{dv}{dt} = \sum T_c, \]  
(6)

where \( m \) is solid mass, \( v \) is translational velocity, \( I \) is inertia tensor, \( w \) is rotational velocity, \( F_c \) is contact force, \( F_d \) is drag force, \( F_b \) is buoyancy, \( g \) is acceleration of gravity and \( T_c \) is torque contact [12]. Furthermore, the contact force has a normal and tangential component, which is represented in Equation (7).

\[ F_c = F_{cn} + F_{ct}, \]  
(7)

where \( F_{cn} \) is normal contact force and \( F_{ct} \) is tangential contact force, consequently the contact torque is equivalent to the tangential torque, as shown in Equation (8).

\[ T_c = T_T, \]  
(8)

where \( T_c \) is contact torque and \( T_T \) is tangential torque; from what Cundall and Strack (1979) postulate, the normal and tangential contact forces act on the particles, generating the movement through the fluid. Now, to evaluate the contact forces, the model can be linearized, using a spring-damper type model [12] making use of different coefficients to studied particle associated with friction, damping and stiffness, thus obtaining Equation (9) and Equation (10).

\[ F_{cn} = -k_n \delta_n - n_n v_n, \]  
(9)

\[ F_{ct} = -k_t \delta_t - n_t v_t, \]  
(10)

where \( k_n \) is the stiffness, \( k_t \) is the tangential stiffness, \( g_n \) is the normal damping coefficient, \( g_t \) is the tangential damping coefficient, \( \delta_n \) is the displacements between particles, \( \delta_t \) is in the normal and tangential directions, \( v_n \) is normal velocity and \( v_t \) is tangential velocity. The particles sliding is related to Coulomb friction model [13] which is defined as follows (see Equation (11)). Therefore, by replacing in Equation (8), we obtain Equation (12).

\[ F_{ct} = -\frac{\mu F_{cn} \delta_t}{|\delta_t|}, \]  
(11)

\[ T_T = r_n \ast F_{ct}, \]  
(12)

where \( r_n \) is the normal vector, \( n_n \) is the radius of the particles.

2.2.4. Fluid motion model. The finite volume method is used to carry out the analysis in addition; the Navier-Stokes equations are used to describe the water flow [14]. The corresponding Equation (13) and Equation (14) are shown below.

\[ \frac{\partial}{\partial t} (\varepsilon p_t) + \frac{\partial}{\partial x_1} (\varepsilon p_t u_1) = 0, \]  
(13)
where \( p_f \) is the density of the fluid, \( u \) is the velocity of the water, \( p \) is the pressure of the water, \( \mu \) is the viscosity of the water, \( x \) is the denotation of coordinates and the interaction between the forces of buoyancy and \( F_s \) is drag [13,14], \( \varepsilon \) is defined as the specific porosity in the calculation of an empty cell [14], as shown in Equation (15).

\[
\varepsilon = 1 - \sum_{i=1}^{N} \frac{V_{p,i}}{V_{cell}},
\]

where \( N \) is the total number of particles present in a specific cell of the CFD, \( V_{p,i} \) is the volume of particle and \( V_{cell} \) is the volume of the cell in the CFD, [15]. In this way, the model for the movement of the flow is proposed, contemplating the inherent physical aspects and the guarantee of the dragging of the particles [16].

### 2.3. Processing (simulation conditions)

In this phase of the simulation development, the solution of the previously proposed mathematical model is carried out. In the first place, the precision is assigned to the solution, to be obtained, in this case, the simple solution monitor is activated, given the model low complexity, and an average solution of 100 iterations is also executed [17], at the end, the data obtained during the sizing and 3D model phase are entered for the particles (velocity and diameter) [18].

### 2.4. Post-processing

The results are organized in tables, and the visualization of the model is made from a deformed geometry, in this way the flow lines, the particles behavior (sand) and other relevant data can be clearly visualized [19].

### 3. Results and discussions

The results obtained from the simulation, present several aspects and emergent physical phenomena within each proposed unit, ST, with the hydraulic simulation it is possible to visualize and analyze them to a great extent, however, to determine the highest efficiency and comply with the objective of study, the analysis is limited to velocity and behavior of the sand.

#### 3.1. Flow velocity

The flow velocity inside each ST has been affected by different factors associated with the structure, its geometry, the type of screens and their order; velocity flow is a fundamental aspect in each ST since it guarantees the drag of the solids and the constant flow inside the ST [1].

Figure 1 shows the flow behavior inside each proposed unit, ST; how in the STs the velocity presents variations, due to the flow that occurs in a chaotic and turbulent way, these phenomena also induce the potential energy gain, in the flow and the appearance of vorticity, which in turn produces dead zones in the ST, (see Figure 1(a), Figure 1(c), Figure 1(d), Figure 1(e), and Figure 1(f)) [2], these high and irregular velocities make the flow remain for a short time in the ST and do not guarantee a good settling of the solids. Figure 1(b) shows how the flow passes in order due to the proposed diffusers, so there will be no physical phenomenon that affects hydraulic operation [3].

Figure 2 shows how effectively the ST with double perforated screen presents a slight variation in velocity, while the ST without perforated screens present the most abrupt velocity variations [4]; however, the ST that have a single perforated screen also varies significantly, in contrast to the unit that has two perforated screens [5].
Figure 1. Flow velocity lines for (a) smooth screens, (b) perforated screens, (c) interleaved smooth-perforated screens, (d) Interleaved perforated-smooth screens, (e) ST of a single smooth screen, and (f) ST of a single perforated screen.

Figure 2. XY velocity graphs from (a) smooth and perforated double screen units, and (b) single and interleaved screen units.

3.2. Behavior of the sands inside each unit
The sands behavior and their transit to the interior of each ST is affected by the emergent physical phenomena and the hydraulic operation of the same, and this in efficiency turn conditions of the STs and their ability to settle solids [6,7]. It can be seen in the Figure 3 that the only ST that presents an ordered flow from the proposed diffusers is the one with a double perforated screen, on the contrary, the other ST present within themselves, turbulent flows, vorticity and dead zones, this influences in a way negative in the ST hydraulic operation since it generates solids resuspension, when they are trapped in the vortex and not find an orderly flow, they tend to go back to the surface, this is evidenced by the efficiency, which does not exceed 50%, while in the ST double perforated screen it is approximately 78% [8].

3.3. Stability of each model from courant number
The courant number is used to determine the model stability; the courant number guarantees stability when your result is close to one [9,10]. Table 1 shows the results obtained for each model as a function of the length (x) and the courant number.
From the values obtained, the analyzed models’ stability is evidenced, since the courant number in most of them remains above 0.5 [19], however, there are small unstable areas in the models with double interleaved screen, in which there is a Courant number lower than 0.5, however, its influence on the general stability is minimum [19].

**Figure 3.** Sands behavior inside each ST for (a) sands in the smooth screen grit trap, (b) sands in the perforated screen grit trap, (c) ST of interleaved smooth-perforated screens, (d) ST of Interleaved perforated-smooth screens, (e) ST of a single smooth screen, and (f) ST of a single perforated screen.

**Table 1.** Courant number for each unit, based on length (x).

| X (m) | # Courant (Smooth double screen) | # Courant (Perforated double screen) | # Courant (Smooth perforated screen) | # Courant (Perforated perforated screen) | # Courant (Single smooth screen) | # Courant (Single perforated screen) |
|-------|----------------------------------|--------------------------------------|--------------------------------------|-----------------------------------------|----------------------------------|-------------------------------------|
| 1     | 0.99820                          | 0.99931                              | 0.93268                              | 0.95241                                 | 0.88631                          | 0.91477                             |
| 2     | 0.97892                          | 0.99927                              | 0.92007                              | 0.95009                                 | 0.90845                          | 0.93301                             |
| 3     | 0.95207                          | 0.99910                              | 0.91588                              | 0.92144                                 | 0.90001                          | 0.94600                             |
| 4     | 0.94071                          | 0.99475                              | 0.91006                              | 0.92004                                 | 0.92471                          | 0.92401                             |
| 5     | 0.93006                          | 0.99108                              | 0.88375                              | 0.90407                                 | 0.77325                          | 0.86247                             |
| 6     | 0.90010                          | 0.98375                              | 0.68402                              | 0.84152                                 | 0.60028                          | 0.70007                             |
| 7     | 0.84063                          | 0.93018                              | 0.62379                              | 0.72560                                 | 0.59325                          | 0.62989                             |
| 8     | 0.80488                          | 0.97266                              | 0.38541                              | 0.63848                                 | 0.84368                          | 0.89224                             |
| 9     | 0.89177                          | 0.99117                              | 0.24901                              | 0.49005                                 | 0.95884                          | 0.96001                             |
| 10    | 0.97114                          | 0.99901                              | 0.56911                              | 0.40792                                 | 0.88211                          | 0.90106                             |

4. **Conclusions**

The importance of this study is based on correcting traditional dimensioning through computational fluid dynamics model, to avoid low efficiencies in projects in which this type of structures is implemented since, having a ST operating with low efficiency, there will be greater turbidity, which implies that a higher coagulant concentration is used in the treatment plant, this represents a high cost in the process and can generate a deficit in the companies in charge of carrying out the treatment. The ST that presented the best efficiency was the double perforated screen (78%), the other ST proposed did not even exceed 50% efficiency. Based on what was obtained, it is recommended to first place at least two consecutive perforated screens in the ST, in order to obtain high removal efficiencies and better hydraulic performance, it is also suggested that for future studies the model be contrasted, with a model real, which provides more accurate and adjusted data.
References
[1] Alghurabi A, Mohyaldinna M, Jufara S, Younis S, Abduljabbara A, Azuwana M 2020 CFD numerical simulation of standalone sand screen erosion due to gas-sand flow Journal of Natural Gas Science and Engineering 10(103) 103
[2] Cundall, P A, Strack, O D L 1979 A discrete numerical model for granular assemblies Geotechnique 29 47
[3] Chang S, Wang X 2003 Multi-dimensional courant number insensitive Euler solvers for applications involving highly nonuniform 39th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Huntsville: American Institute of Aeronautics and Astronautics)
[4] Zhou L, Zhao Y 2020 CFD-DEM simulation of fluidized bed with an immersed tube using a coarse-grain model Chemical Engineering Science Journal 231 20
[5] Maddahian R, Asadi M, Farhanieh B 2012 Numerical investigation of the velocity field and separation efficiency of deoiling hydrocyclones Pet. Sci. 9 511
[6] Batista D 2019 Mesh-independent streamline tracing Journal of Computational Physics 401 108
[7] Cajias A, Villablanca A 2014 Diseño de un desarenador para el pre filtrado de agua de riego Informativo INIA Urri. 98 1
[8] Ratish B, Naidu K 1993 Hydrodynamics of bubble column reactors at high gas velocity: experiments and computational fluid dynamics (CFD) simulations Industrial & Engineering Chemistry Research 46(25) 8431
[9] Parsi M, Agrawal M, Srinivasan V, Vieira R E, Torres C F, McLaury B S, Shirazi S A 2015 CFD simulation of sand particle erosion in gas-dominant multiphase flow Journal of Natural Gas Science and Engineering 27(2) 706
[10] Shirzadi M, Parham Mirzaei P, Naghashzadegan M 2017 Improvement of k-epsilon turbulence model for CFD simulation of atmospheric boundary layer around a high-rise building using stochastic optimization and Monte Carlo Sampling Technique Journal of Wind Engineering and Industrial Aerodynamics 171 366
[11] Saidi M, Maddahian R, Farhanieh B, Afshin H 2012 Modeling of flow field and separation efficiency of a deoiling hydro cyclone using large eddy simulation International Journal of Mineral Processing 112(113) 84
[12] Wang J, Guo F, Zhao Q, Wu M 2013 Numerical simulation of flow field in underground cyclone desander with different entrances Advanced Materials Research 655(657) 470
[13] Gorobets A, Tarabara V, 2020 Separation performance of desanding and deoiling hydrocyclones treating three-phase feeds: effect of oil-particle aggregates Separation and Purification Technology Journal 237 10
[14] Liang X, He L, Luo X, Li Q, You Y, Xu Y 2021 Experimental study on sand particles accumulation, migration and separation efficiency in slug catcher Chinese Journal of Chemical Engineering 32 134
[15] Siaw K, Yung J, 2019 CFD study of sand erosion in pipeline Journal of Petroleum Science and Engineering 176 269
[16] Awad A M, Hussein I A, Nasser M S, Karami H, Ahmed R, 2020 CFD modeling of particle settling in drilling fluids: impact of fluid rheology and particle characteristics Journal of Petroleum Science and Engineering 199 108
[17] Fukada T, Takeuchi S, Kajishima T 2014 Effects of curvature and vorticity in rotating flows on hydrodynamic forces acting on a sphere International Journal of Multiphase Flow 58 292
[18] Jeong W, Seong J 2014 Comparison of effects on technical variances of computational fluid dynamics (CFD) software based on finite element and finite volume methods International Journal of Mechanical Sciences 78 19
[19] De Plaza J, Pulgarin., Ruge J, Rojas J 2021 Hydraulic modeling of combined sewers overflow integrating the results of the SWMM and CFX models Rev. Int. Métodos Numér. Cálç. Diseño Ing. 37(1) 14